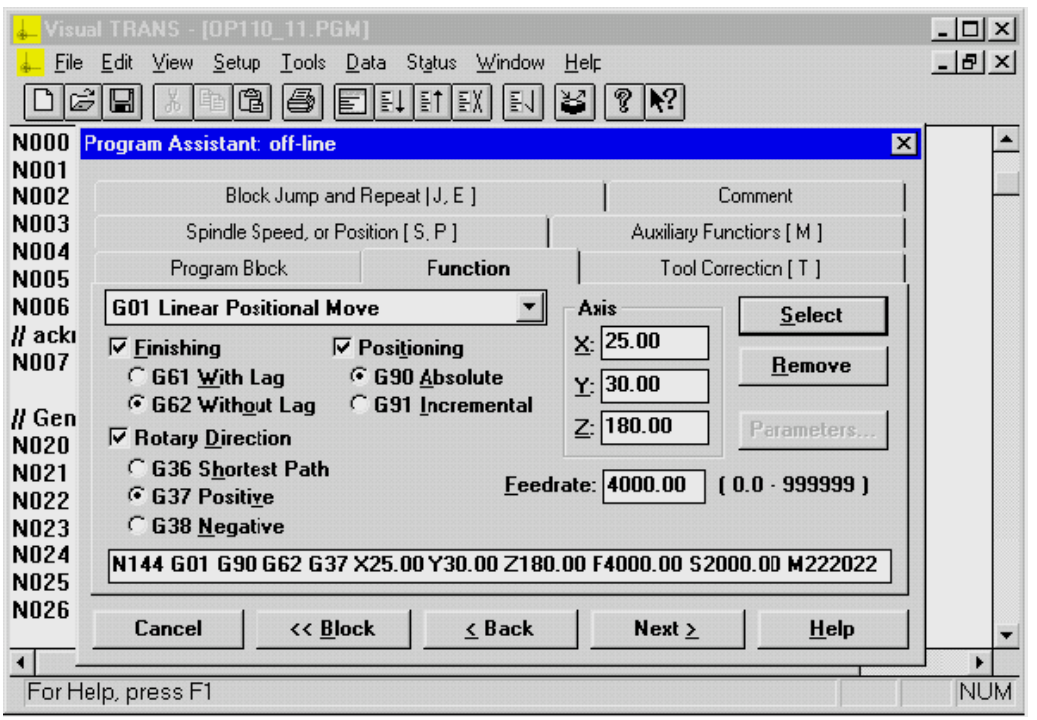
engineering
mannesmann
Rexroth



Visual TRANS 07VRS
Motion Control Software

User Manual

DOK-CONTRL-VT*PC-07VRS-AW01-AE-P

Rexroth
Indramat

Title
Visual TRANS 07VRS
Motion Control Software

Type of Documentation
User Manual

Document Typecode
DOK-CONTRL-VT*PC-07VRS-AW01-AE-P

Internal File Reference
- Publication number: 120-0400-B327-01/AE

Purpose of Documentation
This documentation describes the installation and use of the Wndows based Visual TRANS motion control software that can be used to program the TRANS 01-D.

Record of Revisions

Description	Release Date	Notes
01	05/00	Initial Release

Copyright
© 2000 Rexroth Indramat GmbH
Copying this document, giving it to others and the use or communication of the contents thereof without express authority, are forbidden. Offenders are liable for the payment of damages. All rights are reserved in the event of the grant of a patent or the registration of a utility model or design (DIN 34-1).

Validity
All rights are reserved with respect to the content of this documentation and the availability of the product.

Published by
Rexroth Indramat GmbH • Bgm.-Dr.-Nebel-Str. 2 • 97816 Lohr am Main • Germany • Tel.: 09352/40-0 • Telex: 689421 • Fax: 09352/40-4885
Rexroth Indramat Division • 5150 Prairie Stone Parkway • Hoffman Estates, IL 60192 • USA • Tel.: 847-645-3600 • Fax: 847-645-6201
http://www.rexroth.com/indramat
Dept. ESV3

Table of Contents

1 Introduction to Visual TRANS 1-1

1.1 Installing Visual TRANS ... 1-1
1.2 Running Visual TRANS ... 1-5
1.3 Creating a Program Block ... 1-7
Program Block Tab ... 1-8
Function Tab ... 1-8
Tool Correction [T] Tab ... 1-11
Spindle Speed, or Position [S, P] Tab ... 1-12
Auxiliary Functions [M] Tab ... 1-13
Block Jump [J] Tab ... 1-14
Comment Tab ... 1-15
1.4 Saving an NC Program ... 1-16
1.5 Menu Bar ... 1-16
File Menu ... 1-16
View Menu ... 1-17
Setup Menu ... 1-18
Tools Menu ... 1-23
Data Menu ... 1-34
Status Menu ... 1-37
1.6 Tool Bar ... 1-42
Program Assistant ... 1-42
Program Download… ... 1-43
Program Upload… ... 1-43
Clear NC Program… ... 1-43
Check Program Syntax ... 1-43
Archive… ... 1-43
About Visual TRANS… ... 1-43
Help ... 1-43

2 CLC DDE SERVER 2-1

2.1 Dynamic Data Exchange ... 2-1
The Dynamic Data Exchange Server ... 2-1
Dynamic Data Exchange Interface ... 2-1
2.2 The Communication Servers Main Window ... 2-2
Settings Menu - CLC Server Configuration ... 2-3
Settings Menu - Serial Communications ... 2-4
2.3 SERVER Topic Name ... 2-7

3 Programming 3-1

3.1 Application Programming Requirements ... 3-1
Start of the Program ... 3-1
End of the Program ... 3-1
3.2 Programming Capability Description ... 3-2
3.3 Recommended Programming Styles with an Example ... 3-2
Automatic Mode ... 3-2
Manual Mode ... 3-4
3.4 TRANS 01-D NC Code Descriptions ... 3-7
Axis Enable and Disable (G20, G21) ... 3-8
Basic Homing Program ... 3-8
Homing and Zero Offset (NC Code G74 & G69) ... 3-9
Positioning (NC Code G00, G01, G90 & G91) ... 3-12
With / Without Lag During Positioning (G61 & G62) ... 3-13
Enable/Disable Feed To A Positive Stop (G75 & G76) ... 3-14
Adaptive Depth Control (G08) ... 3-15
External Feedback Devices - Distance Coded Linear Scale ... 3-19
Rotary Motion Control ... 3-20
Rotary Axis examples - Feedrate Interpretation ... 3-24
Feedrate (NC Code F) ... 3-26
Dwell (NC Code G04) ... 3-26
Tool Corrections (NC Code T) ... 3-26
Spindle Speed Control (NC Code S) ... 3-30
Spindle Positioning Control (NC Code P) ... 3-30
Auxiliary Functions (NC Code M) ... 3-32
Program Jumps ... 3-33

4 CLC DRIVE PARAMETER EDITOR 4-1

4.1 Overview ... 4-1
4.2 Parameters Menu ... 4-1
Analog Outputs ... 4-1
Drive Direction ... 4-2
Drive Name ... 4-3
Drive Monitoring ... 4-3
Drive Tuning ... 4-4
Drive Limits ... 4-5
Drive Reference ... 4-6
Encoder 2 ... 4-7
Mechanical ... 4-9
Overview ... 4-10
4.3 Oscilloscope ... 4-12
File Menu ... 4-12
Source Menu ... 4-12
Timing ... 4-13
Signal Selection ... 4-13
Options menu ... 4-15

Abort, Upload and Enable Trigger 4-17
Oscilloscope memory buttons 4-17
Manipulating trace signals 4-18
Time Controls 4-18

5 CLC TRANSFER UTILITY 5-1

5.1 Contents for CLC_XFER Server Help 5-1
5.2 The Communication Servers Main Window 5-1
5.3 Dynamic Data Exchange Interface 5-1
To transfer data to or from a file 5-1
To get status or the last error message 5-1
To change a parameter in the server 5-2
5.4 Activity Log 5-2

6 Index 6-1

7 Customer Service Locations 7-1

1 Introduction to Visual TRANS

Visual TRANS is based on the current TRANS 01-D Programming Worksheet. It allows the TRANS 01-D to be programmed from a laptop computer through screen prompts within a Microsoft® Windows™ environment. Visual TRANS also allows saving, and editing any of the 200 *NC Program Blocks* (N000 - N199), which can include jumps from a current NC program block to any of the other 199 NC program blocks.

This manual describes the use of Visual TRANS. Tool bars, drop-down menus, and variables are all described, and the step-by-step process of creating an NC program block using Visual TRANS is also discussed.

Use of Visual TRANS will be intuitive to users familiar with Microsoft® Windows™. For users unfamiliar with Microsoft® Windows™, the following information also contains detailed instructions with illustrations that mirror what the user will see on the computer as they use Visual TRANS.

1.1 Installing Visual TRANS

Install the Visual TRANS program on the computer using the following method.

1. From Windows Explorer (Windows 95 / Windows 98 / Windows NT), double-click (rapidly press the left mouse button twice) on the *Setup.exe* file. Another option from Start (Windows 95 / Windows 98 / Windows NT) is to left-click on ***Run...*** and type the complete file path and file name and left-clicking on the ***OK*** button.

Note: If this installation is being done from a floppy disk, then select the appropriate drive letter. (A: or B: is typically a floppy drive designation.)

After executing Setup.exe, using one of the above methods, the following Windows 95 screen will appear.

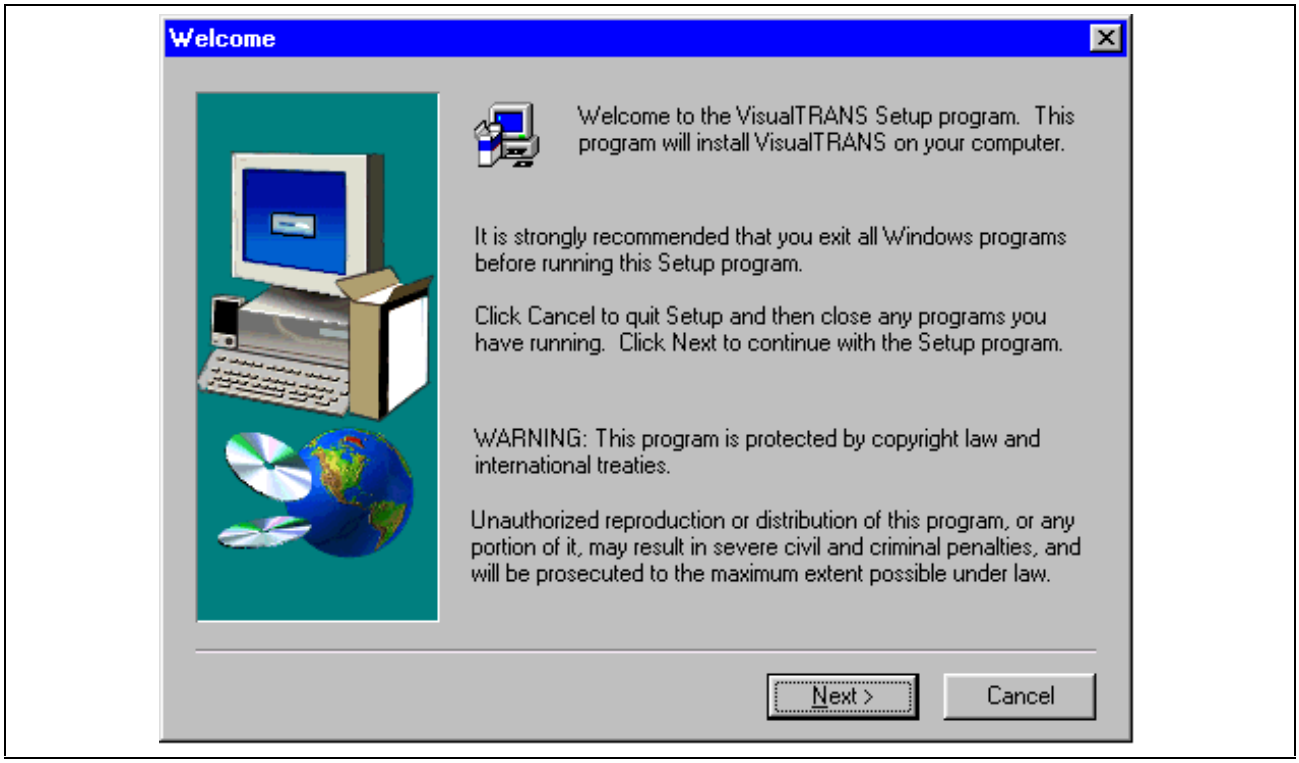

Welcome
Welcome to the VisualTRANS Setup program. This program will install VisualTRANS on your computer.
It is strongly recommended that you exit all Windows programs before running this Setup program.
Click Cancel to quit Setup and then close any programs you have running. Click Next to continue with the Setup program.
WARNING: This program is protected by copyright law and international treaties.
Unauthorized reproduction or distribution of this program, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the maximum extent possible under law.
Next >
Cancel

Fig. 1-1: First Screen in Setup.exe

Note: To select an item, move the mouse to position the pointer directly on the item to be selected. Press the left mouse button to select or "click" on it.

To end the installation of Visual TRANS at any time, select ***Cancel***.

2. Click the ***Next>*** button to reveal the following screen.

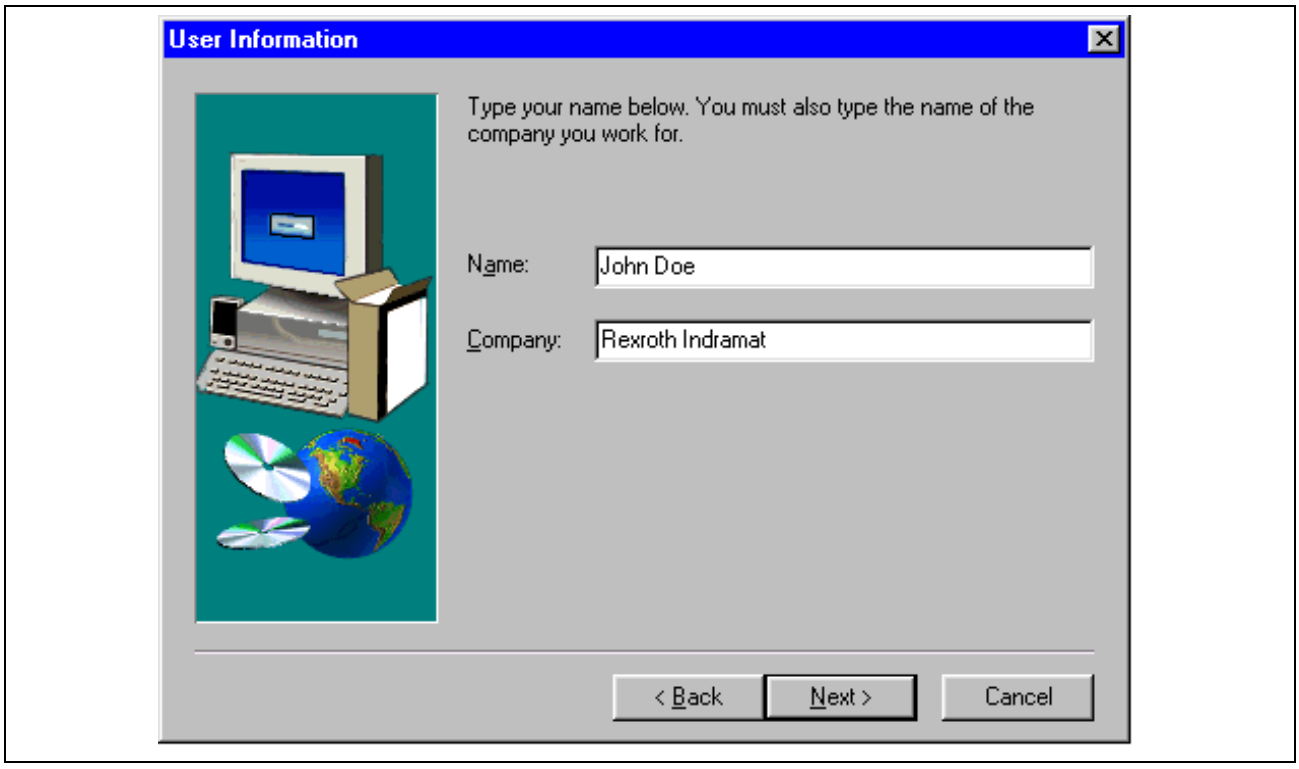


Fig. 1-2: Second Setup.exe Screen

3. Enter a user name and a company name; the installation cannot continue without either value being entered. After the user name and company name are input, select ***Next>*** to continue. If there is a need to return to the previous screen, click ***<Back***.

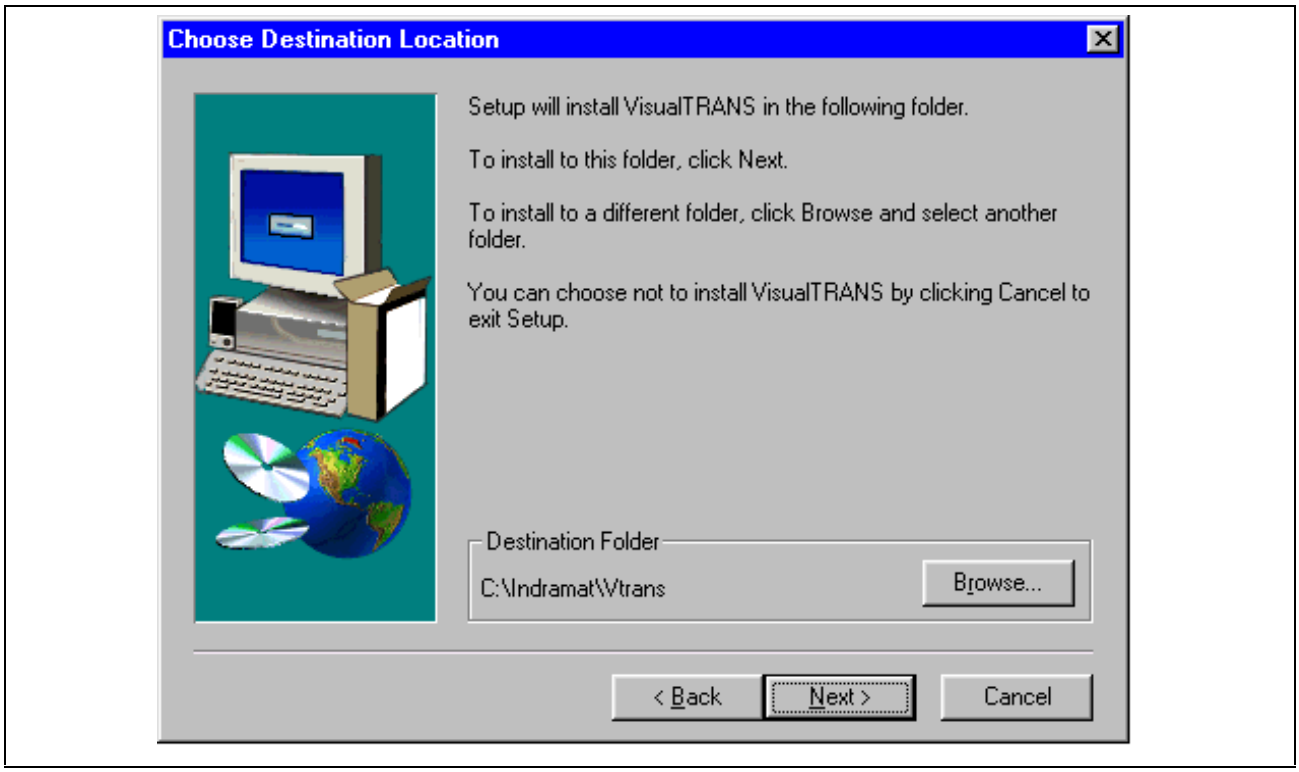

Choose Destination Location
Setup will install VisualTRANS in the following folder.
To install to this folder, click Next.
To install to a different folder, click Browse and select another folder.
You can choose not to install VisualTRANS by clicking Cancel to exit Setup.
Destination Folder
C:\Indramat\Vtrans
Browse...
< Back
Next >
Cancel

Fig. 1-3: Setup.exe Destination/Location Screen

4. If desired, you may select a destination for the Visual TRANS files on the computer's hard drive that differs from the installation program's default location (*C:\Indramat\Vtrans*) by clicking ***Browse...***. If not, simply click ***Next>*** to continue the installation.

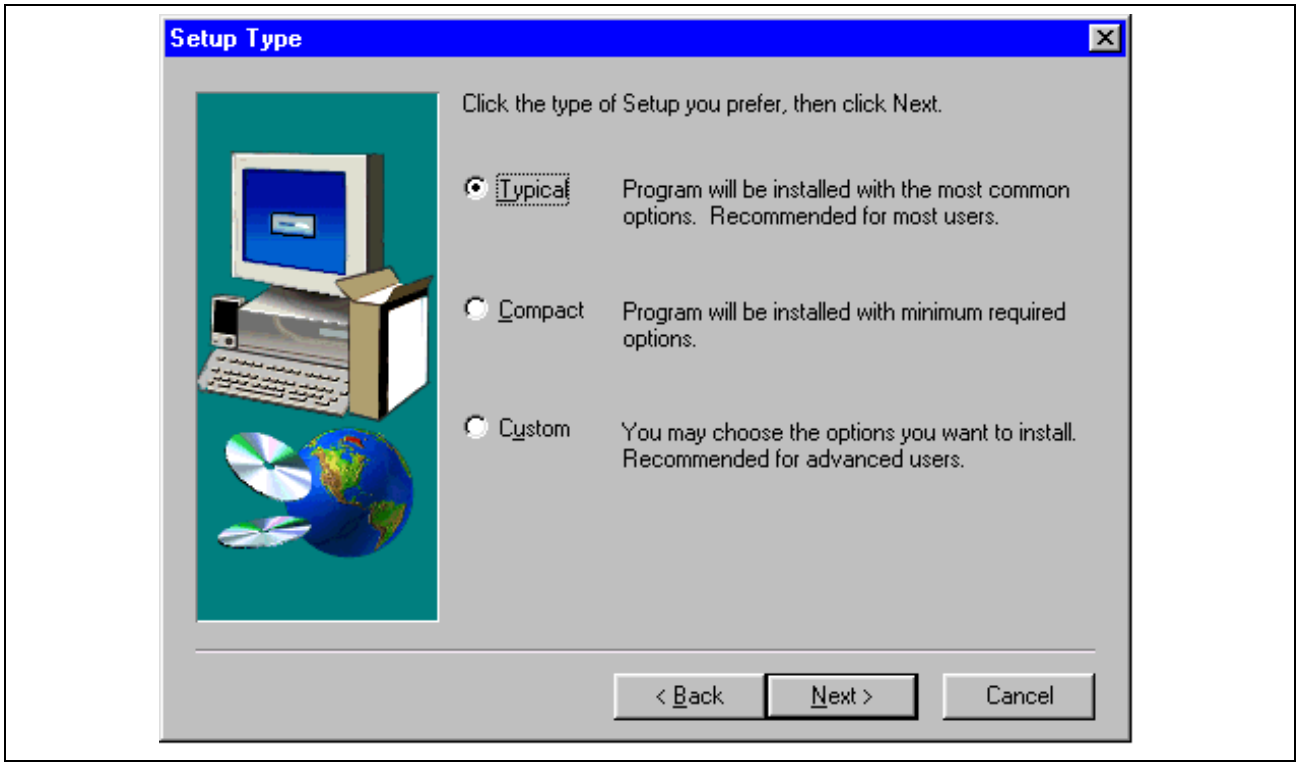

Setup Type
Click the type of Setup you prefer, then click Next.
Typical
Program will be installed with the most common options. Recommended for most users.
Compact
Program will be installed with minimum required options.
Custom
You may choose the options you want to install. Recommended for advanced users.
< Back
Next >
Cancel

Fig. 1-4: Setup.exe Setup Type Screen

5. Choose which setup type will be used by positioning the mouse pointer directly over the appropriate option button and clicking on it.

The default setup type is **Typical**. Click ***Next>*** to continue the installation.

Note: If the setup type of **Custom** is chosen, then the **Select Components** screen will come up after the ***Next>*** button is clicked on. Select the components to be installed and clear the components that should not be installed. After the components have been selected/cleared, click ***Next>*** to continue the installation.

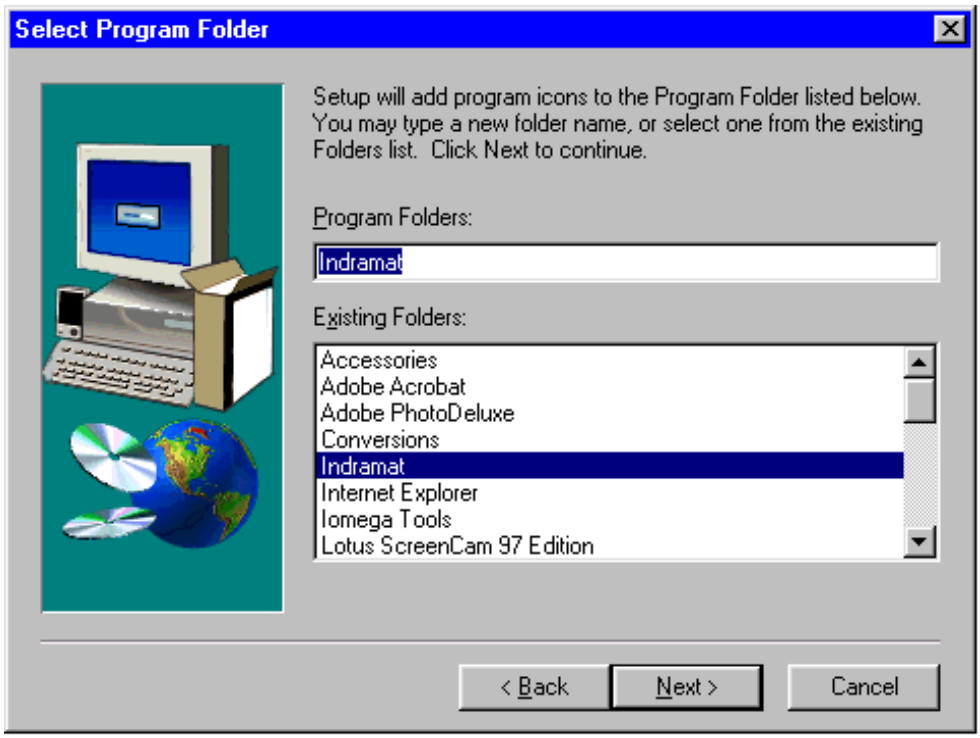


Fig. 1-5: Setup.exe Select Program Folder Screen

6. If desired, you may name the program group that contains the icons generated by the Setup.exe installation. Otherwise, the installation program will assign the name **Indramat**. Select ***Next>*** to continue.

   Visual TRANS will display a status bar that shows the installation progress and waits for the user to select ***Finish*** on the following screen.

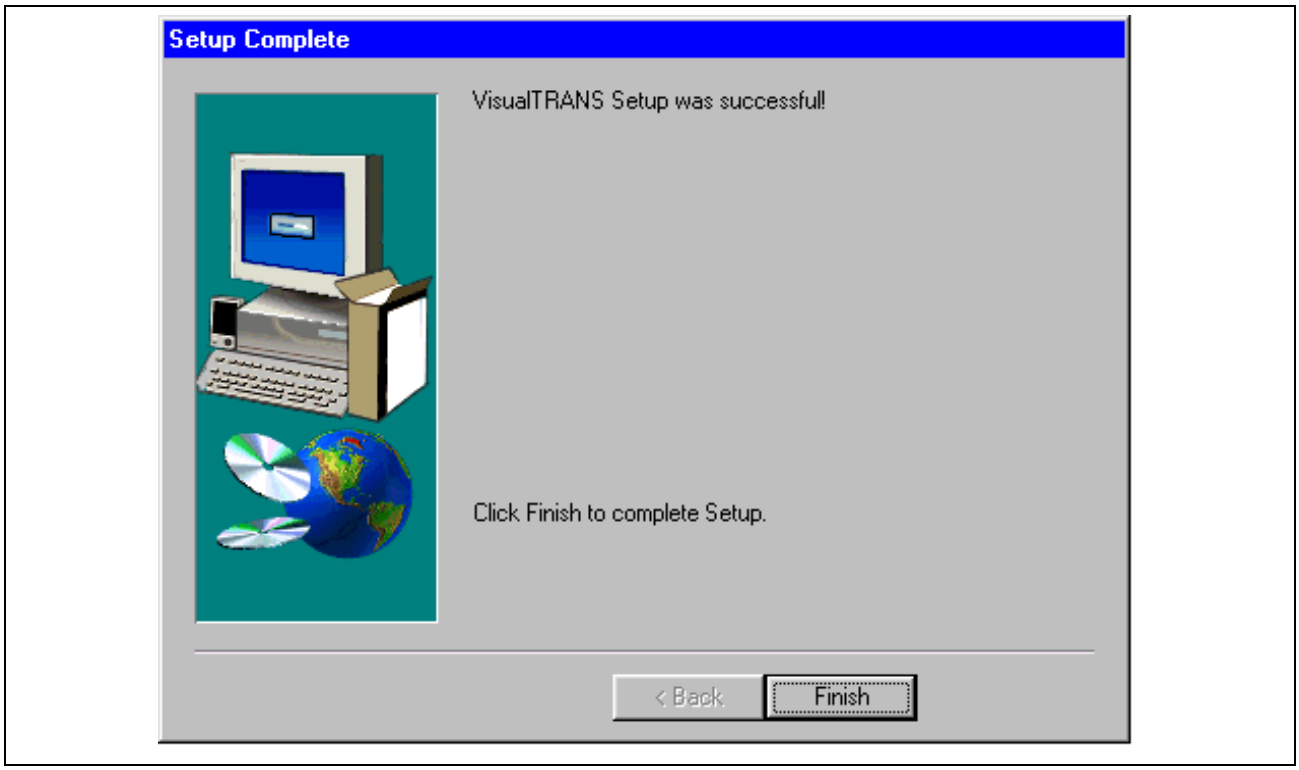


Fig. 1-6: Setup.exe Setup Complete Screen

7. Click ***Finish*** to complete the installation of Visual TRANS.

1.2 Running Visual TRANS

From Windows Explorer (Windows 95 / Windows 98 / Windows NT), double-click on the directory called **Indramat**. Next, double-click on **Vtrans**. Lastly, to begin the Visual TRANS application, double-click on the **Vtrans.exe** file.

With Windows 95 / Windows 98 / Windows NT, another method that will utilize the installed icons is as follows:

1. Double-click on “My Computer.”
2. Double-click on the root directory select during setup. Example; “[*C:*]”
3. Double-click on **Indramat**.
4. Double-click on the **Vtrans** folder that appears.
5. Double-click on the Vtrans.exe icon.
6. If you wish to poll the network to identify the available CLC-D cards (when applicable), then click on the ***Yes*** button of the **Visual TRANS** window that is displayed (see Fig. 1-7 below). At this point the **Select Card** window will be opened and the network will be automatically polled (refer to Fig. 1-8 below). After the polling has completed, either (i) click on one of the *Available Cards*, and then click on the ***OK*** button or (ii) click on the ***Begin Polling…*** button in order to poll the network again (refer to Fig. 1-9 below).

   If you do not wish to poll the network to identify the available CLC-D cards, then click on the ***No*** button of the **Visual TRANS** window that is displayed (see Fig. 1-7 below).

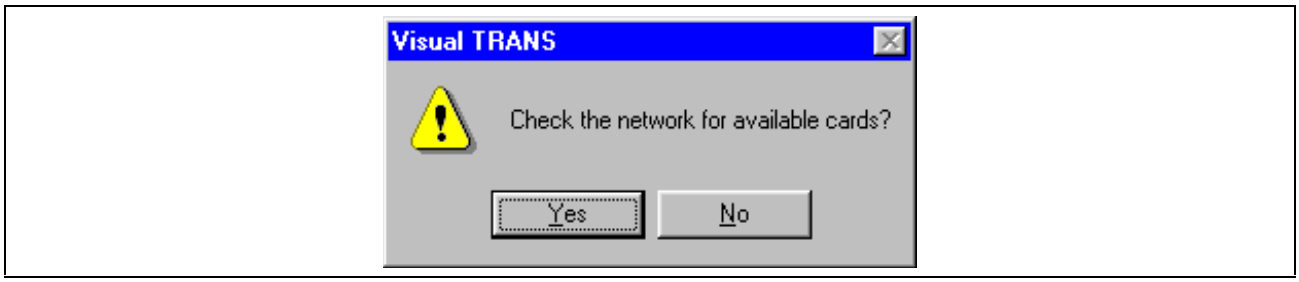


Fig. 1-7: 'Poll the network for CLC cards?' (screen #1)

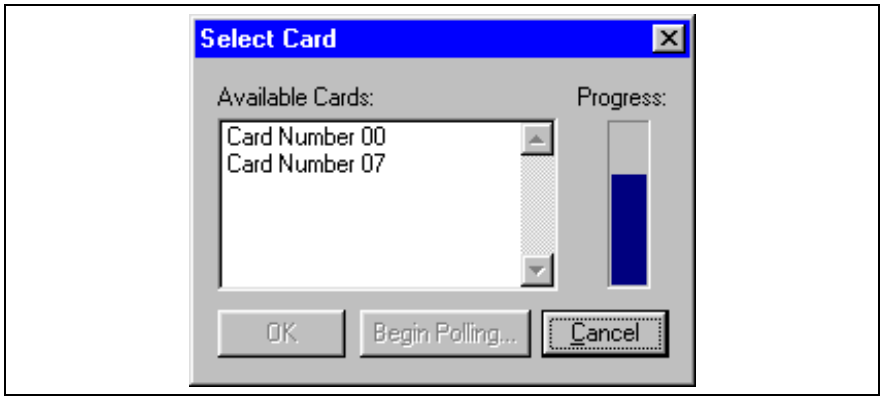


Fig. 1-8: 'Poll the network for CLC cards?' (screen #2)

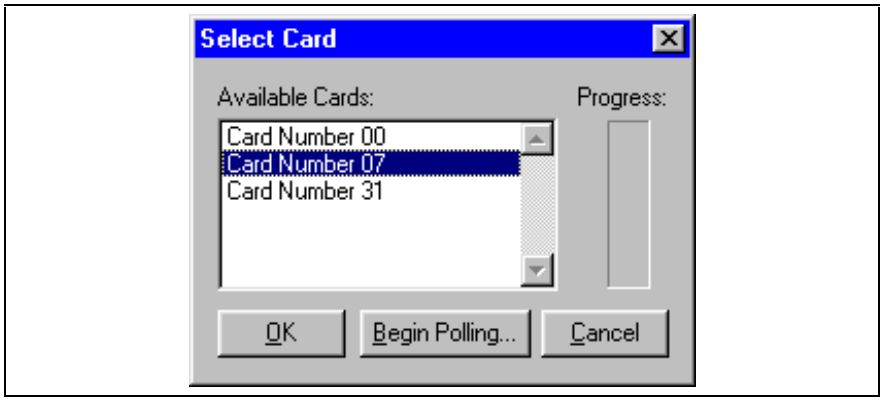


Fig. 1-9: 'Poll the network for CLC cards?' (screen #3)

Upon successful completion of Visual TRANS execution, the following screen will appear as the main screen within the Visual TRANS application.

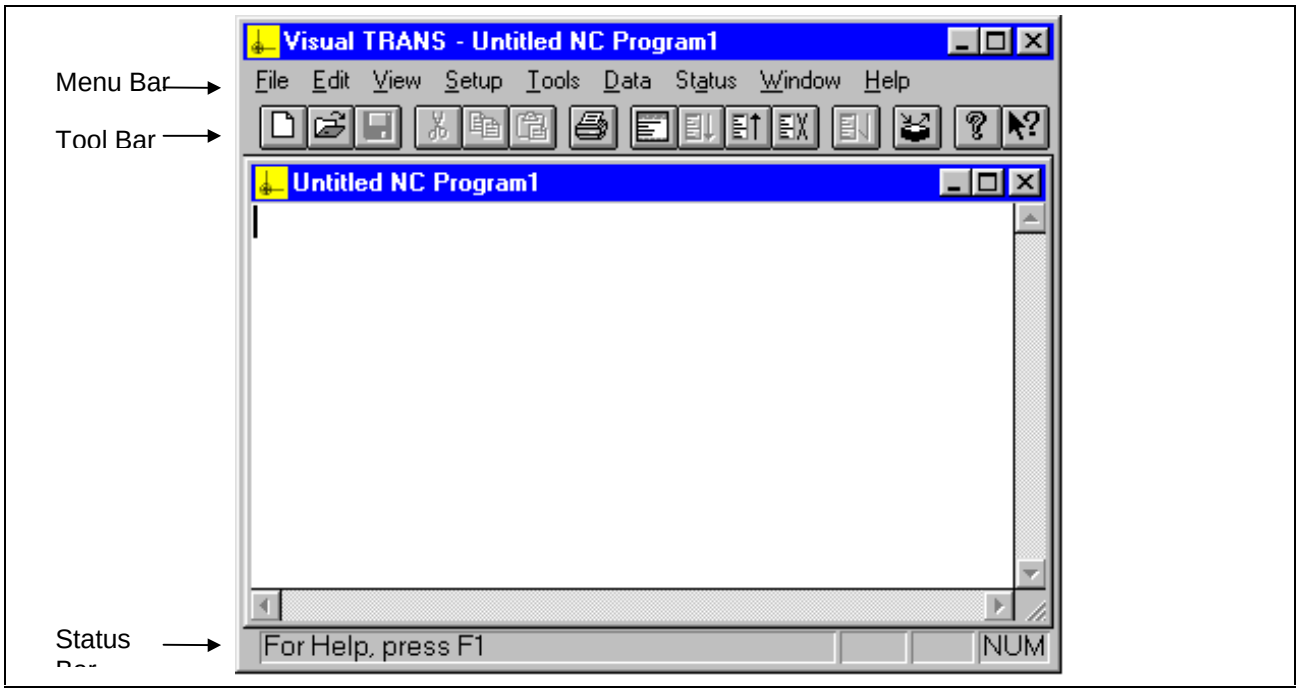


Fig. 1-10: Main screen in Visual TRANS

1.3 Creating a Program Block

To create an NC Program Block, pull-down the **Tools** menu and select **Program Assistant F5**. The **Program Block** tab window will be displayed.

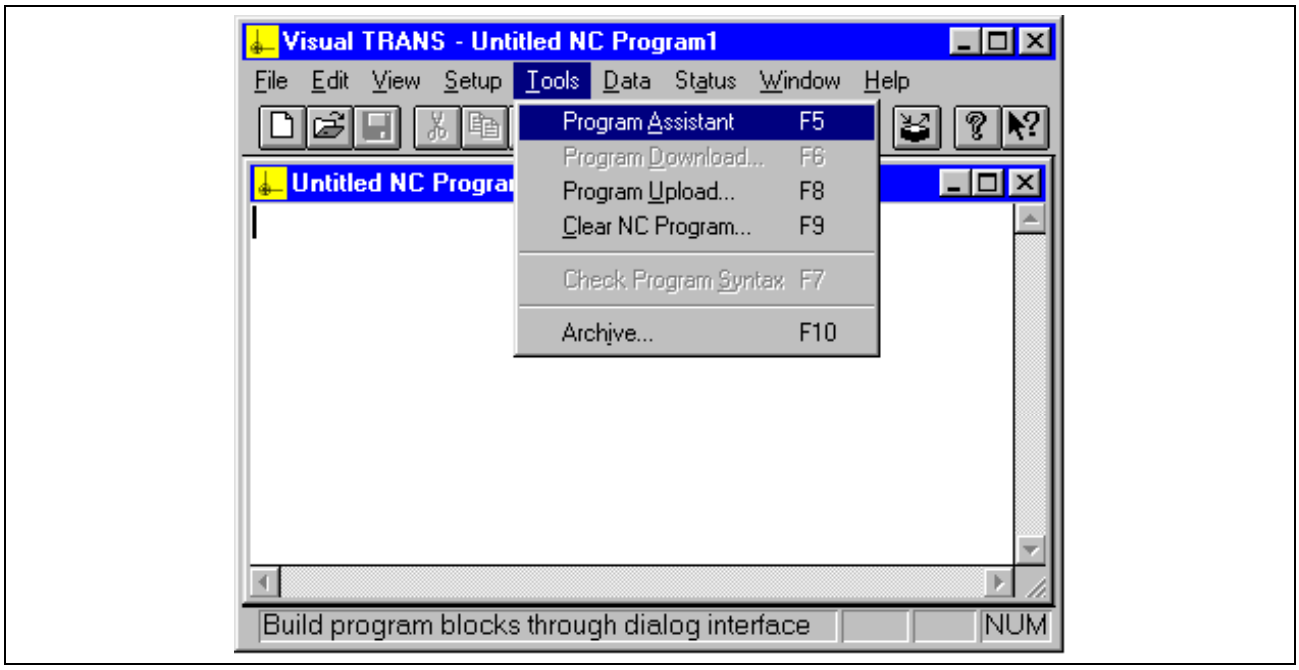

Visual TRANS - Untitled NC Program1
File Edit View Setup Tools Data Status Window Help
Program Assistant F5
Program Download... F6
Program Upload... F8
Clear NC Program... F9
Check Program Syntax F7
Archive... F10
Untitled NC Progra
Build program blocks through dialog interface
NUM

Fig. 1-11: Tools Drop-Down Menu

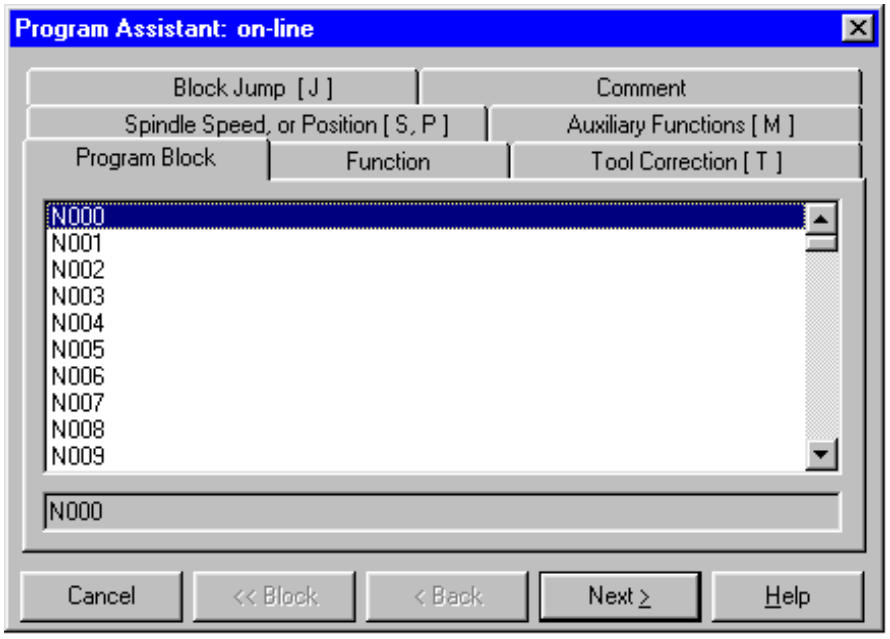


Fig. 1-12: Program Block Tab

This is the first step in creating an NC program block using Visual TRANS. For more detailed information on programming, see **Chapter 3, Programming**.

Program Block Tab

Select the NC program block number to store the commands that the user will input through Visual TRANS. Use the cursor keys on the keyboard to scroll to the desired block number (N###) or use the mouse to select it.

Select ***Next>*** and the **Function** tab of the **Program Assistant** window appears.

Function Tab

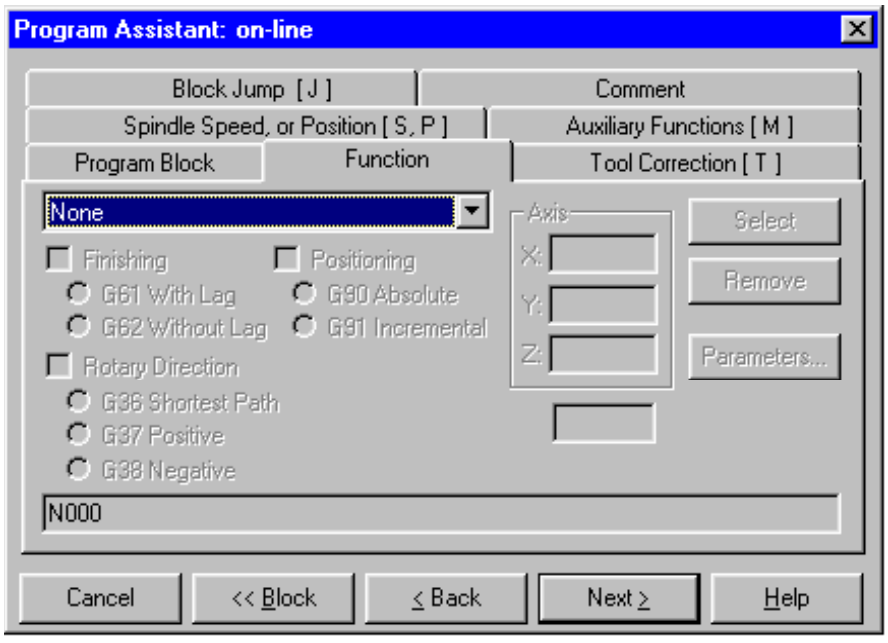


Fig. 1-13: Function Tab of the Program Assistant

Using the mouse, click on the scroll arrow next to "*None*" to display all of the G-Code options that may be selected, as shown in Fig. 1-14.

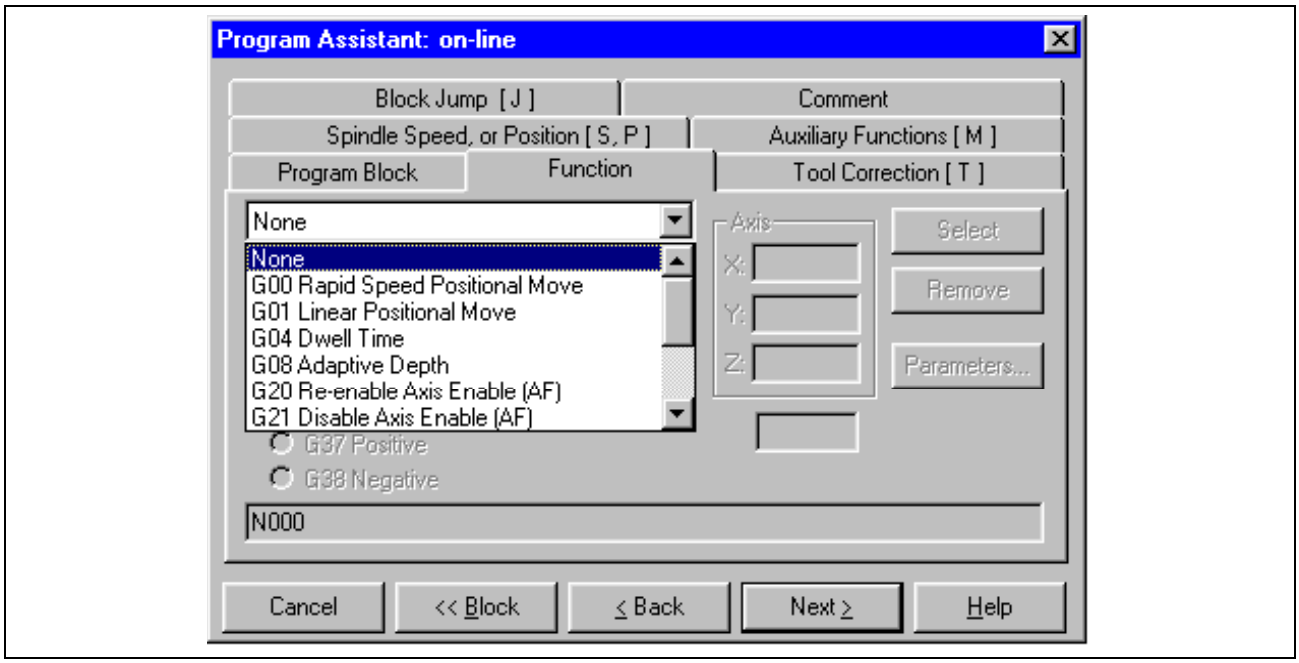


Fig. 1-14: G-code drop-down menu within the Function tab of Program Assistant

Use ↓ (the down arrow key) or click and hold down the left mouse button on the down scroll arrow to move to the bottom of the list at "*Modal Functions (G3x, G6x, G9x, F)*." Click and hold down the left mouse button on the up scroll arrow to move back to the top of the list. When at the bottom of the list, the screen will display the following:

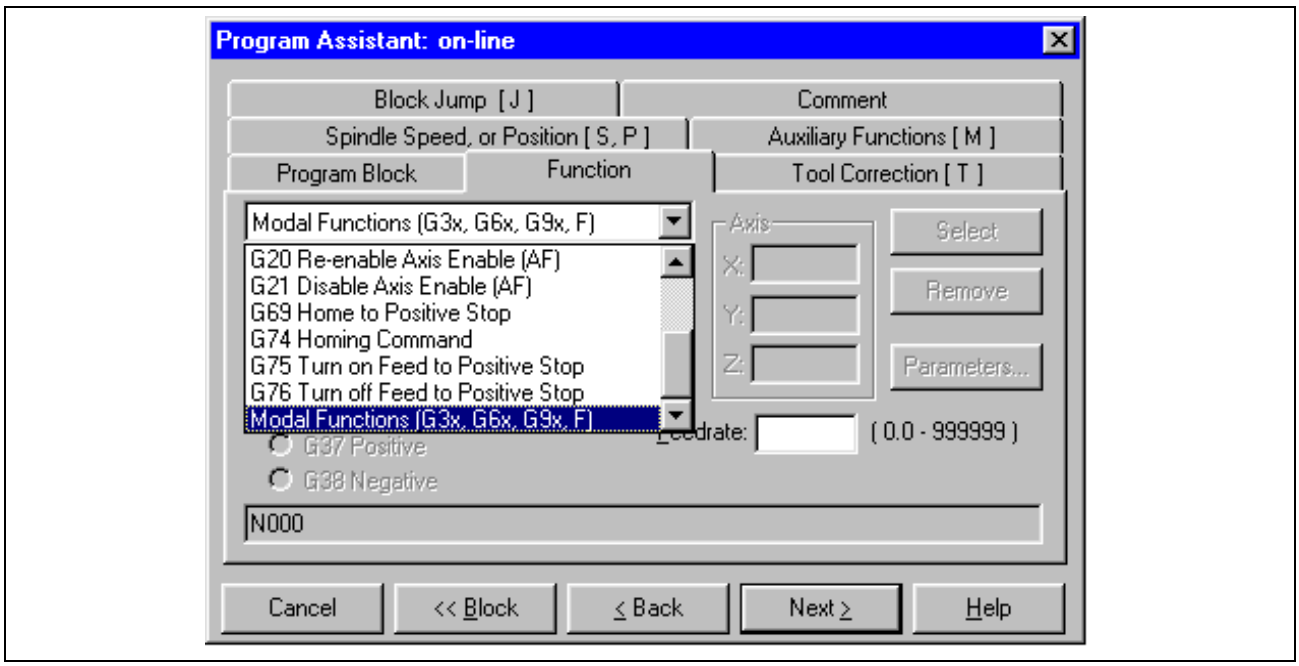
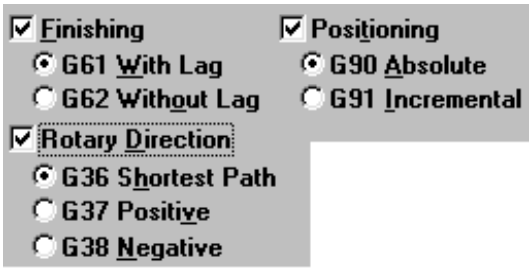


Fig. 1-15: Bottom of the Drop-down menu of the Function tab of Program Assistant

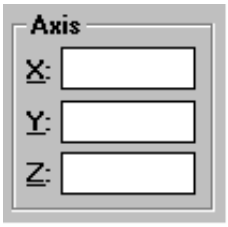
From the drop-down menu choices, select the desired G-code. Use the mouse to select the **Finishing**, **Positioning**, and **Rotary Direction** options.

Note: A check mark in the check box and a black dot next to the **Finishing** and/or **Positioning** and/or **Rotary Direction** options indicates that the option is selected or active. If a check mark is not present, the item is not selected (it is inactive). Change the status by selecting the check box and then selecting the appropriate option button next to the desired **Finishing** or **Positioning** or **Rotary Direction** command.



Enter the *Axis* movements by entering the travel distances in the boxes.

Note: Use the mouse to position the cursor inside the boxes. The arrow becomes an I-beam once located within the white box. Press the left mouse button and then use the keyboard to enter the numeric value. Do this for all of the applicable axes.



Next, enter the feed rate (units/minute. The TRANS 01-D specifies the units (standard inch or metric).

Parameters

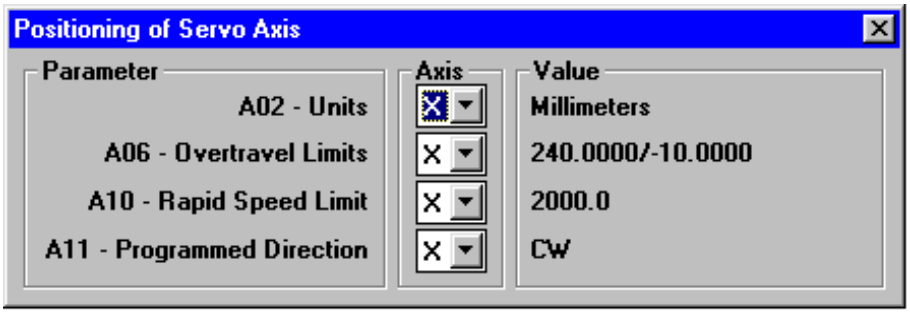
If any of the following G-codes are selected, the ***Parameters...*** option may be selected to view the drive parameters within the context of the selected function. This is only available if Visual TRANS is connected and communicating with a TRANS 01-D.

G00 Rapid Speed Positional Move

G01 Linear Positional Move

G74 Homing Command

When parameters is selected, the following screen will appear.



Click the down scroll arrows in the *Axis* column to select a different axis to view the corresponding *Parameters* and *Values* for that axis.

After all of the information has been selected and the numeric variables have been entered, click on ***Select*** to add them to the NC program block viewing window next to the NC program block number (N###).

Note: The **Positioning of Servo Axis** window will not disappear until the "X" in the upper right-hand corner is selected.

Select ***Next>*** to advance to the **Tool Correction [T]** tab.

Tool Correction [T] Tab

The **Tool Correction** tab will look as follows:

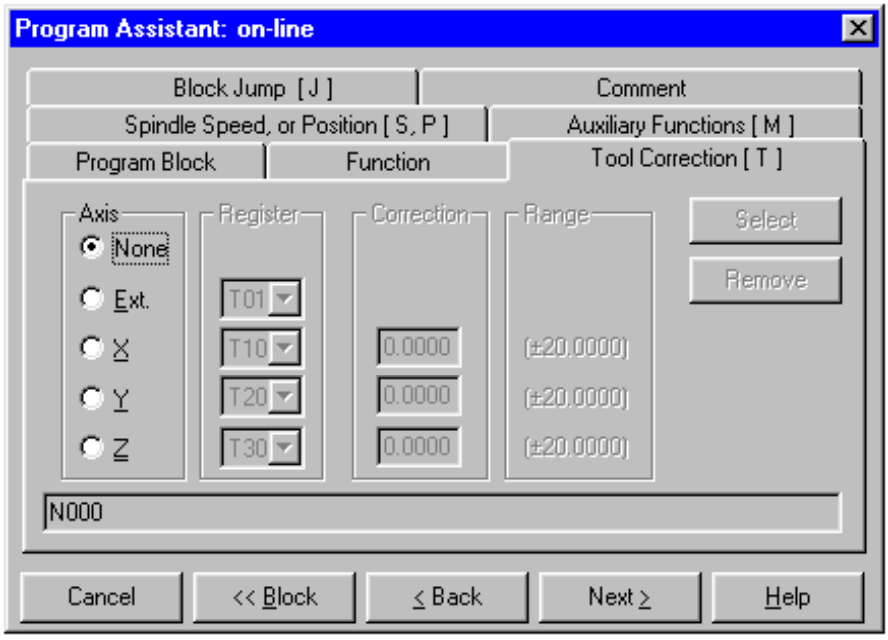


Fig. 1-16: Tool Correction Tab in Program Assistant

Caution

If the TRANS 01-D is on-line and a Tool Correction value is entered, clicking *Select* will IMMEDIATELY download that value for use with the active NC program.

⇒ If the active NC program does not require a Tool Correction, **DO NOT** enter one. Click ***Next>*** to continue to the **Spindle Speed, or Position [S, P]** tab.

Note 1: *Register* numbers ending with “0” will clear that axis offset value.

Note 2: A *Correction* value entered for an axis that is disabled in the TRANS 01-D will create an error message when this NC program block is executed.

Note 3: If the TRANS 01-D is on-line, the *Range* value will be the parameter specified in the TRANS 01-D, and any *Register* values will be displayed upon selection of that axis and *Register* number.

Use of the **Tool Correction [T]** tab is as follows:

1. Choose an *Axis*. (Only one axis is allowed to be selected in each NC program block.)
2. Choose the *Register* number by using the scroll arrows.
3. Enter the *Correction* value, not to exceed the value in the *Range* column.
4. Click ***Select*** to add the tool correction values to the NC program block viewing window.
5. Select ***Next>*** to display the **Spindle Speed, or Position [S, P]** tab.

For detailed information on tool corrections, see **Chapter 3, Programming**.

Spindle Speed, or Position [S, P] Tab

The **Spindle Speed, or Position [S, P]** screen appears as follows:

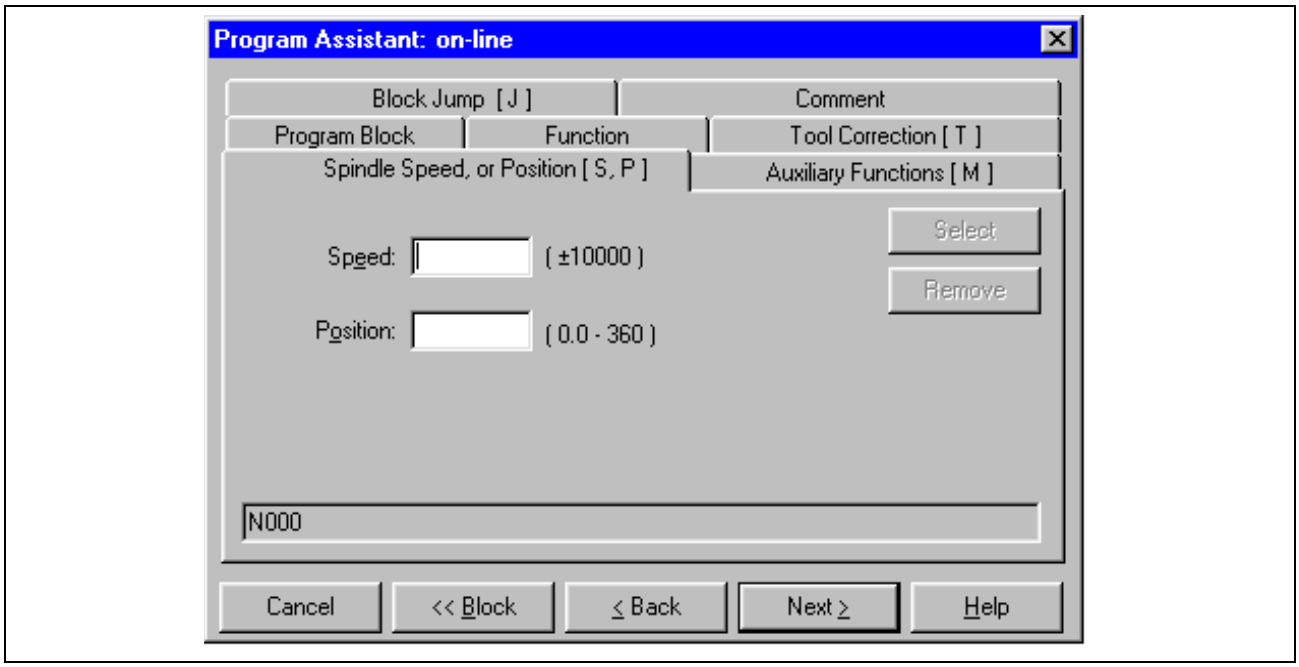


Fig. 1-17: Spindle Speed, or Position [S, P] tab in the Program Assistant

In the appropriate white box, enter the numeric value for **Speed** (Units/minute) or **Position** (Degrees). If a value is entered for either **Speed** or **Position** and the other option is desired, the value must be deleted with the ***Delete*** key or the ***Backspace*** key before Visual TRANS will allow the other option to accept a value.

For more detailed information on spindle settings, see **Chapter 3, Programming**.

Click ***Next>*** to display the **Auxiliary Functions [M]** tab.

Auxiliary Functions [M] Tab

The **Auxiliary Functions [M]** tab will look as follows:

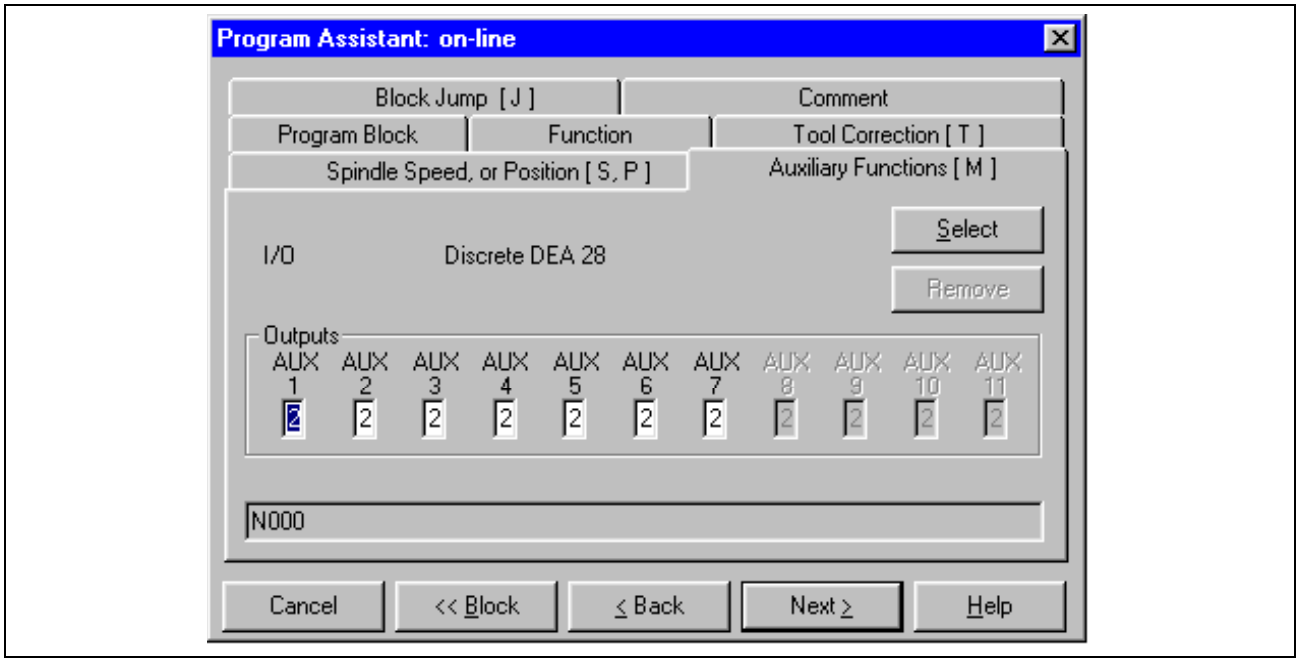


Fig. 1-18: Auxiliary Functions [M] tab in Program Assistant

The Auxiliary Functions [M] options are:

0 = Turn OFF with acknowledgment

1 = Turn ON with acknowledgment

2 = No change

Enter the desired *Auxiliary Output* values and click ***Select*** to add the outputs to the NC program block viewing window.

Click ***Next>*** to display the **Block Jump [J]** tab.

For detailed information on auxiliary functions, see **Chapter 3, Programming**.

Block Jump [J] Tab

The **Block Jump [J]** tab is displayed below:

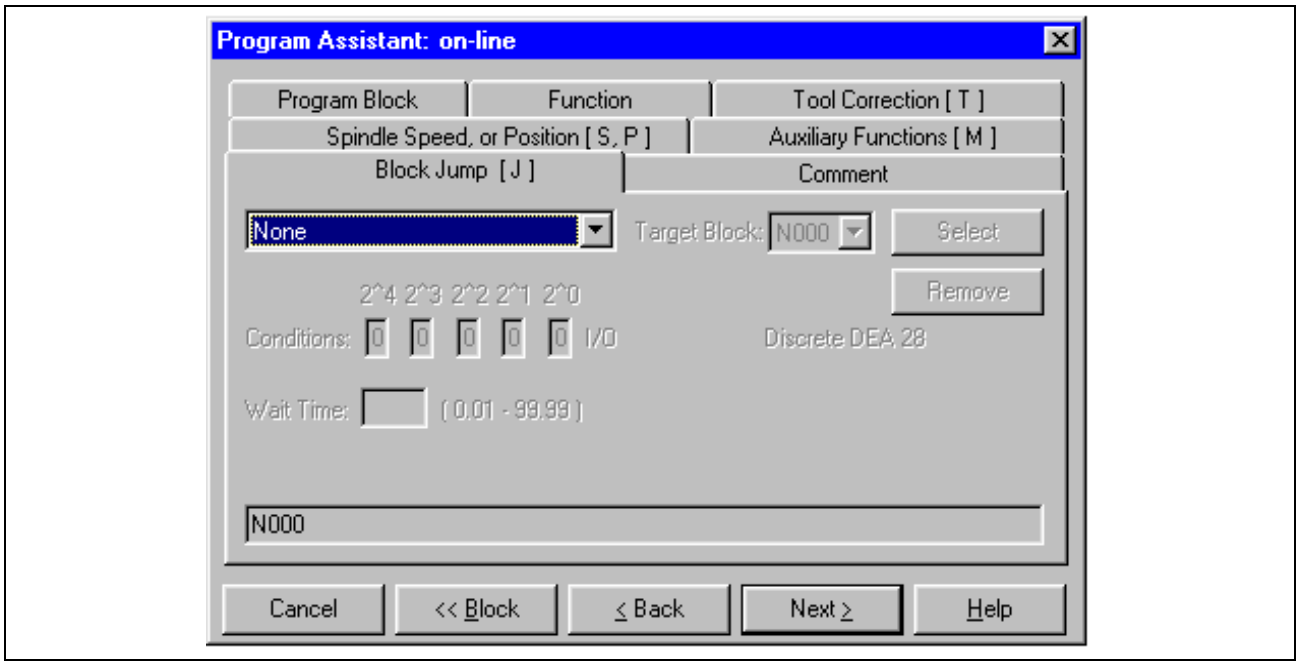


Fig. 1-19: Block Jump [J] tab in the Program Assistant

This screen is used to specify a type of *Block Jump* and the *Block Number* of which the jump is to go to *Target Block*.

Select the scroll arrow next to the "**None**" option to reveal the **Jump** options that may be selected:

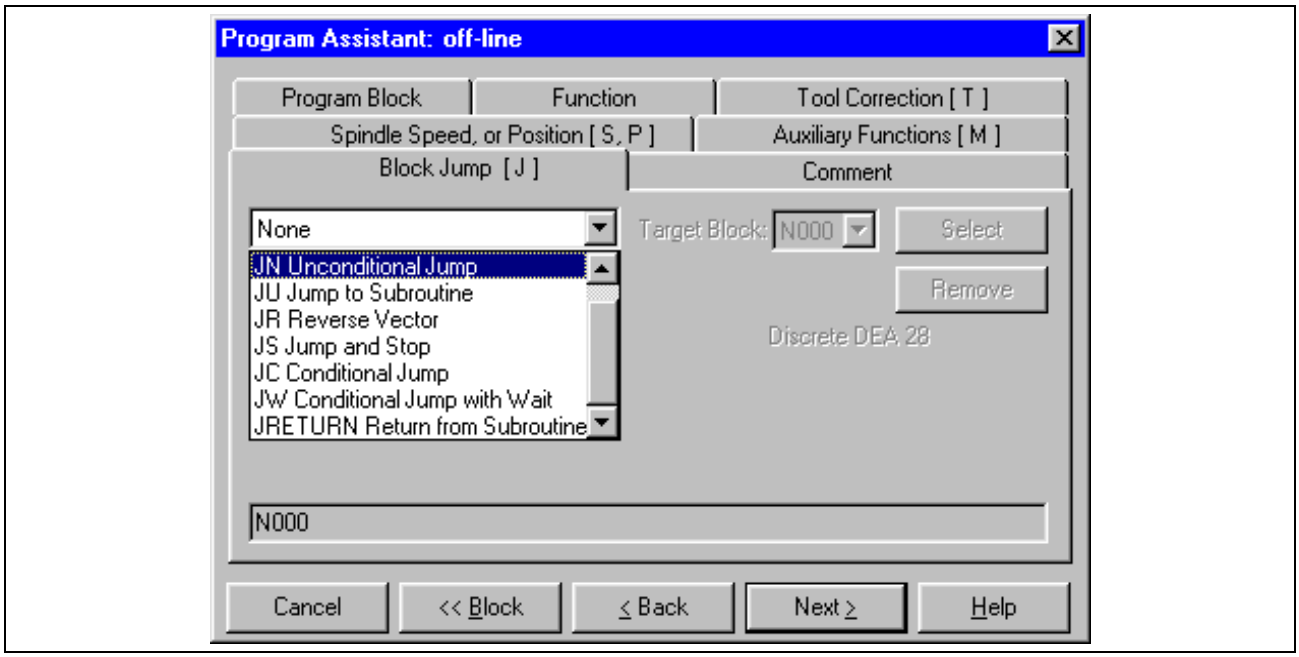


Fig. 1-20: Block Jump [J] tab in the Program Assistant

From the drop-down menu choices, select the desired *Jump Type*. Select the *Target Block* number by using the drop-down menu and selecting the desired *Target Block* number.

If "**JC Conditional Jump**" is selected, the jump *Conditions* must be entered. If "**JW Conditional Jump with Wait**" is selected, the jump *Conditions* and the *Wait Time* (in seconds) must be entered.

Click ***Select*** to add the **Block Jump [J,]** values to the NC **Program Block** viewing window.

Click ***Next>*** to display the **Comment** tab.

For more information on block jumps, see **Chapter 3, Programming**.

Comment Tab

The **Comment** tab allows the user to input a description or information regarding the NC program block. It is useful to identify the purpose of the NC program block and provide a readily available means to determine the function of the NC program block. It will appear as follows:

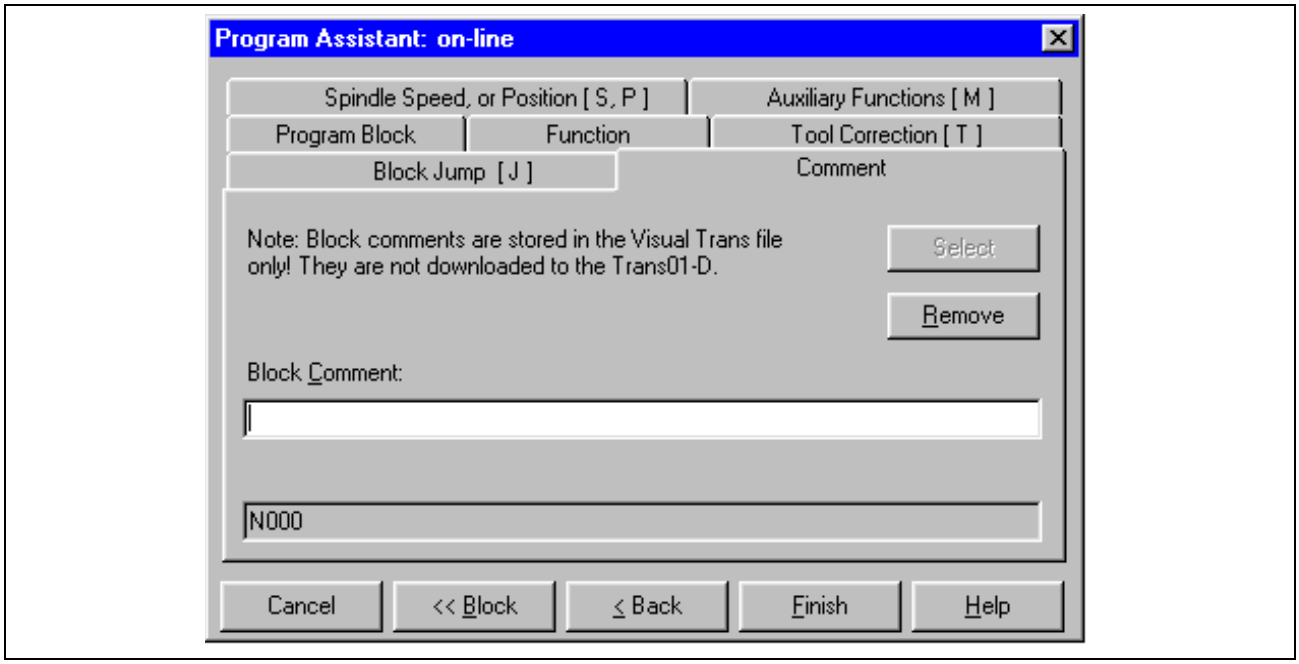


Fig. 1-21: Comment tab in the Program Assistant

Select the white area and input, from the keyboard, the desired *Block Comment* for the NC program block. The maximum length of a comment is 128 characters.

Note: *Block Comments* are saved in Visual TRANS only; they are not sent to the TRANS 01-D with the NC program block.

Click ***Select*** to add the *Block Comment* to the NC **Program Block** viewing window.

Click ***Finish*** to complete the entering of information into the selected NC program block number.

Prior to downloading the NC program to the TRANS 01-D, a check for syntax errors should be made. For information on how to check for syntax errors, see **Check Program Syntax F7** in the **Menu Bar** section.

The last step is to save the NC program that has just been created.

1.4 Saving an NC Program

Saving an NC program allows a user to retain the NC program and use it in future applications.

To save the NC program:

1. Pull-down the **File** menu and select **Save As…**.
2. In the *File name* field, enter the name that will be assigned to the NC program.

Note 1: The NC program name must contain the "***.pgm**" file extension.

Note 2: The NC program should be stored in the *C:\…\Vtrans\Project* subdirectory.

---

3. To finalize the saving of the NC program, click ***OK***. The NC program will be **Saved As** the user-assigned file name.

1.5 Menu Bar

This section will provide information about the menu bar names and the options available in each of the drop-down menus.

Note 1: Several of the menu options available in Visual TRANS are identical to other Microsoft® Windows™ commands. Only menu options that are unique to Visual TRANS are described here. Instructions for using standard Microsoft® Windows™ functions can be found in the Microsoft® Windows™ **Help** menu.

Note 2: Each menu name includes one underlined character. Likewise, each option within each drop-down menu contains one underlined character in its name. These provide an alternative to using the mouse to view drop-down menus and select menu options. To view any drop-down menu, hold down the ***Alt*** key and simultaneously type the appropriate underlined character. To select an option once the menu is displayed, simply type the appropriate underlined character.

File Menu

After selecting **File**, the following drop-down menu will appear.

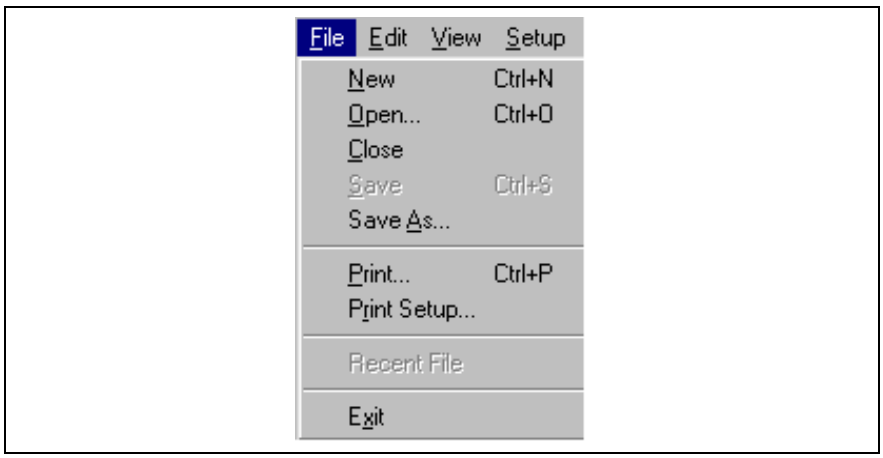


Fig. 1-22: File Drop-Down Menu

New Ctrl+N

This menu item is used to create a **New** file to begin work on a **New** Visual TRANS application. With the selection of **New**, the following window will appear, allowing a user to create a TRANS 01-D program (see the "**Creating a Program Block**" section for more information):

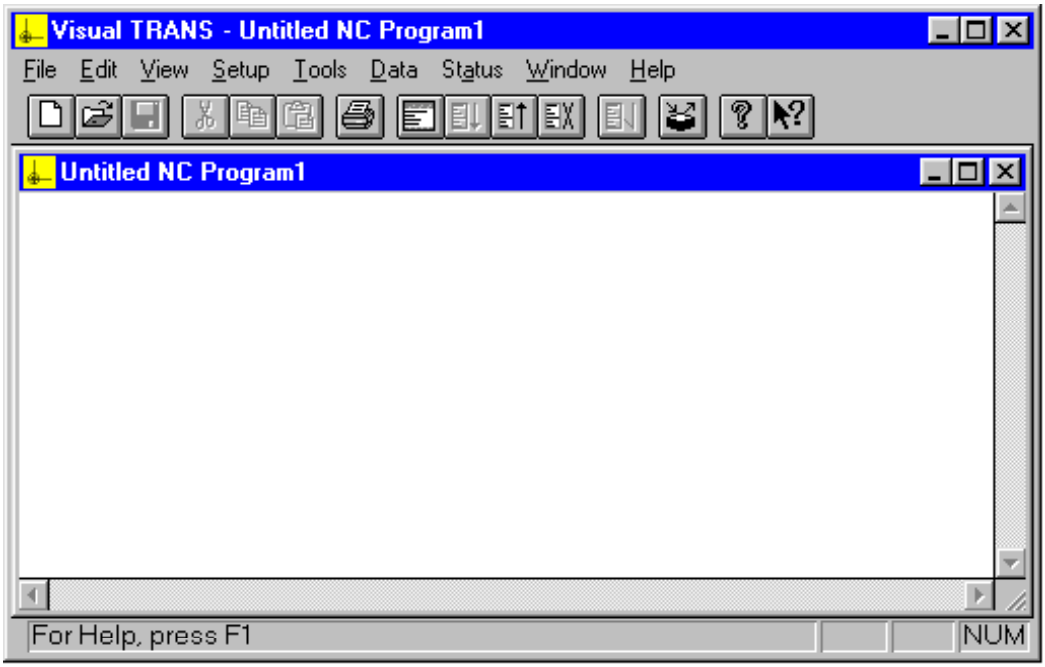

Visual TRANS - Untitled NC Program1
File Edit View Setup Tools Data Status Window Help
Untitled NC Program1
For Help, press F1
NUM

Fig. 1-23: New NC Program Window

Note 1: The appearance of "**Ctrl+N**" represents an alternative method that can be used to select **New** (press the **Ctrl** key and the **N** key simultaneously). This type of selection method is available anytime you see "**Ctrl+_**" next to a menu option.

Note 2: Selecting the **Tool Bar** icon will also execute the **New** command.

View Menu

When selecting the **View** option, the following drop-down menu will appear.

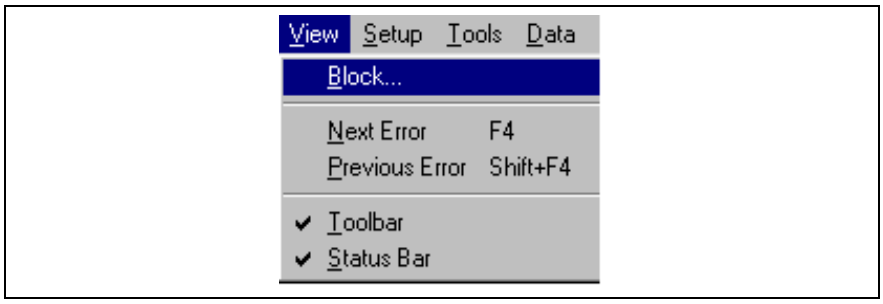
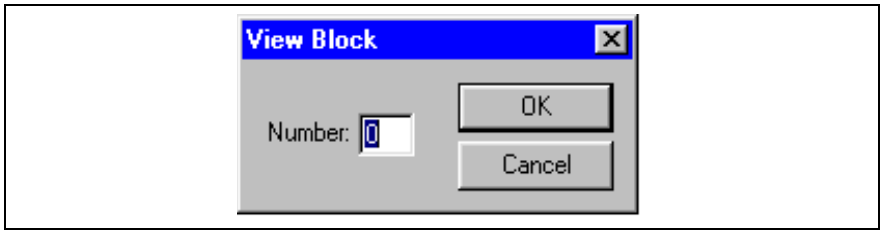


Fig. 1-24: View Drop-Down Menu

Block…



Block allows the user to enter the NC program block number and view the line command contents of the NC block. At the prompt in the white area, enter the NC block number from the keyboard, and select ***OK*** to view the contents of the NC program block.

Next Error F4

When using **Check Program Syntax**, **Next Error** may be used to advance to the next syntax error found in the NC part program if multiple errors are present.

Previous Error Shift+F4

When using **Check Program Syntax**, **Previous Error** may be used to return to a previous syntax error found in the NC part program if multiple errors are present.

Setup Menu

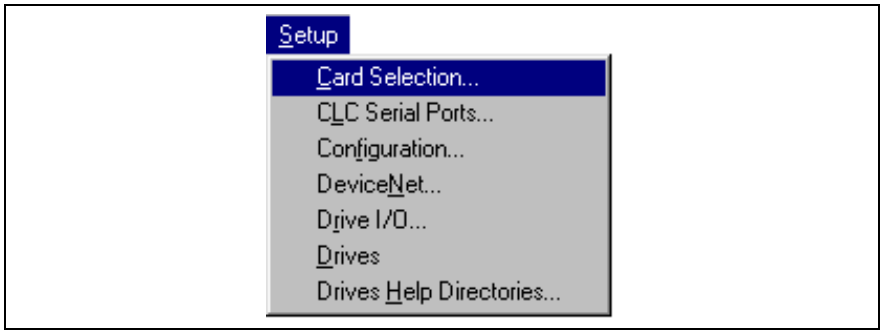


Fig. 1-25: Setup Drop-Down Menu

Card Selection...

Used to specify a method of communication with the TRANS 01-D.

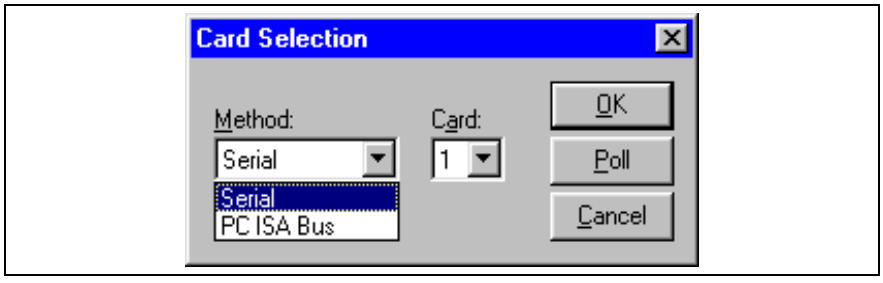


Fig. 1-26: CLC Communication Screen

By clicking on the scroll arrow beneath *Method*, select one of the available methods of CLC communication. Next, enter the *Card* number (0-31).

If you wish to poll the network to identify the available CLC-D cards (when applicable), then click on the ***Poll*** button. At this point the **Select Card** window will be opened and the network will be automatically polled (refer to Fig. 1-27 below). After the polling has completed, either (i) click on one of the *Available Cards*, and then click on the ***OK*** button [the **Select Card**

window will be closed and the *Card* number field will display the card that was selected as a result of the polling] or (ii) click on the ***Begin Polling…*** button in order to poll the network again (refer to Fig. 1-28 below).

When finished, click on the ***OK*** button of the **Card Selection** window.

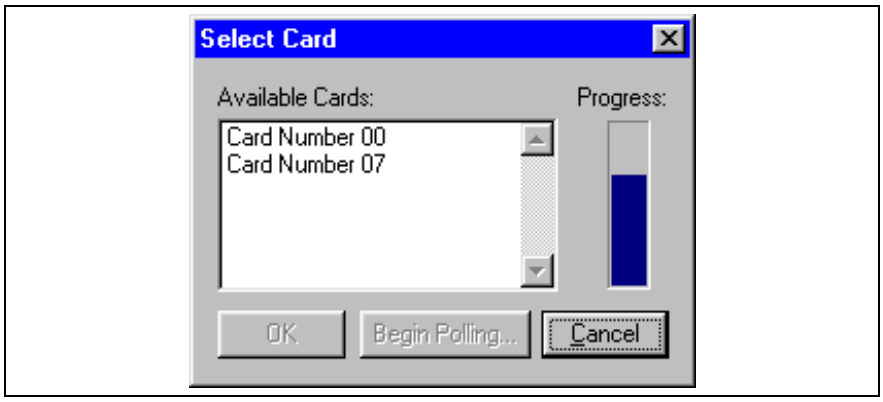


Fig. 1-27: Select Card window (screen shot #1)

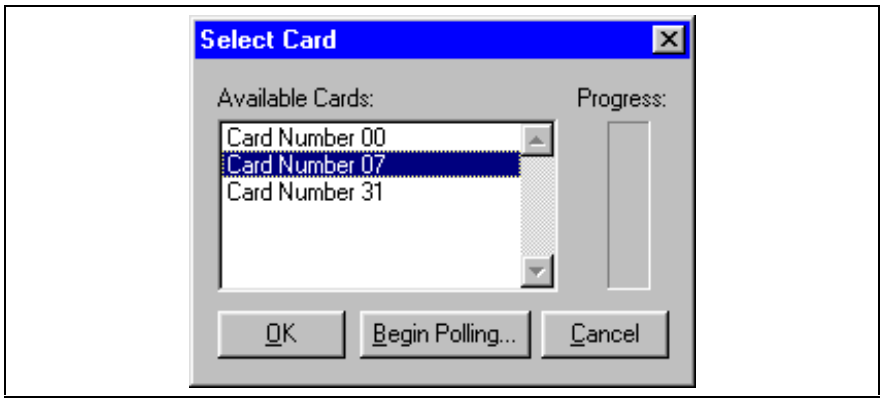


Fig. 1-28: Select Card window (screen shot #2)

CLC Serial Ports...

Used to setup the CLC-D cards two serial ports.

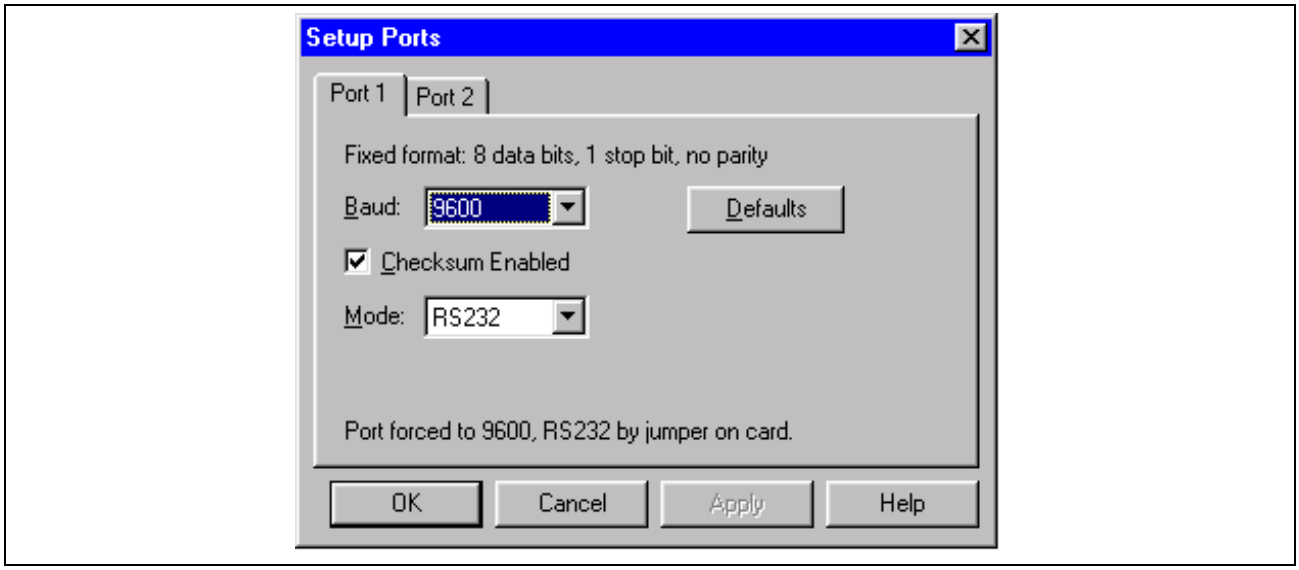


Fig. 1-29: CLC Serial Port 1 Tab

The **Port 1** tab is typically used for communication with the host system and corresponds to serial port X27 on the CLC-D card. The default values for the **Port 1** and **Port 2** tabs are *Baud*: *9600*, *Checksum Enabled*: *Checked*, and *Mode*: *RS232*. These values can be defaulted by clicking on the ***Defaults*** button and then clicking on the ***Apply*** button. **Port 1** is also referred to as **Port A**.

The values for *Baud* and *Mode* can be changed by clicking on the scroll arrow adjacent to each field, selecting a value, and clicking on the ***Apply*** button. It is recommended that the **Checksum Enabled** check box remain checked for both the **Port 1** and **Port 2** tabs.

The *"Port forced to 9600, RS232 by jumper on card"* message is displayed at the bottom of the **Port 1** tab when jumper S1 has been set on the CLC-D card (see Fig. 1-29).

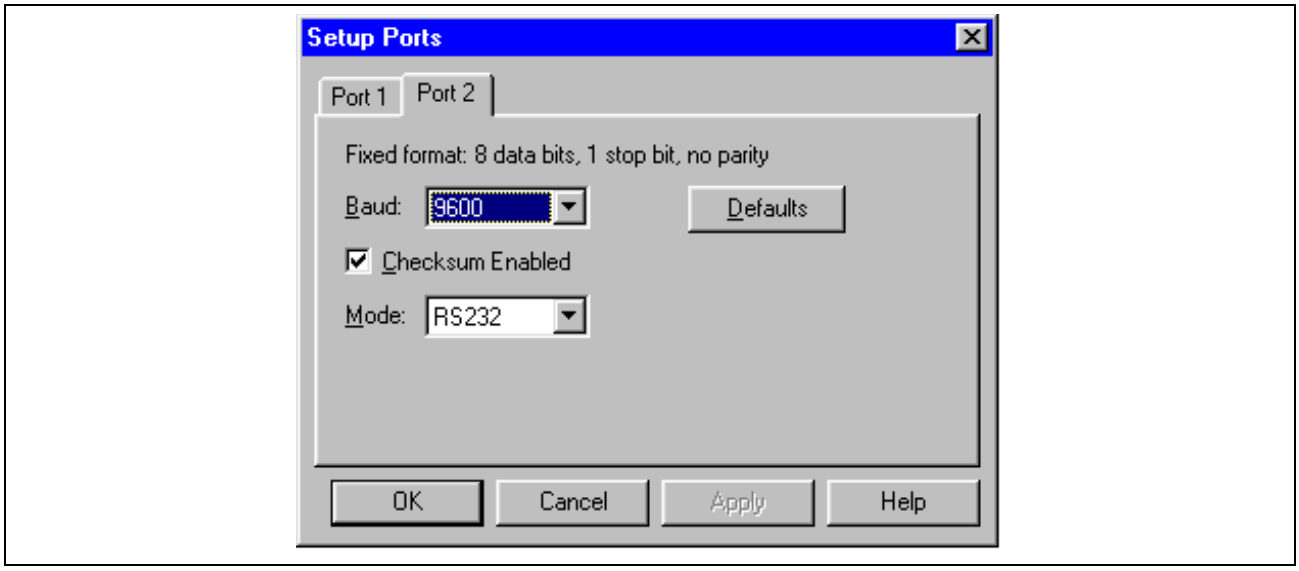


Fig. 1-30: CLC Serial Port 2 Tab

The **Port 2** tab is generally used to communicate with either the CTA 10-1 or BTC06 (when used) and corresponds to serial port X28 on the CLC-D card. *Mode* should be set to *RS232* for the CTA 10-1 and *RS422* for the BTC06. The ***Help*** button is not currently being used. **Port 2** is also referred to as **Port B**.

The *"Port forced to 9600, RS232 by jumper on card"* message is displayed at the bottom of the Port 2 tab when jumper S2 has been set on the CLC-D card.

Configuration

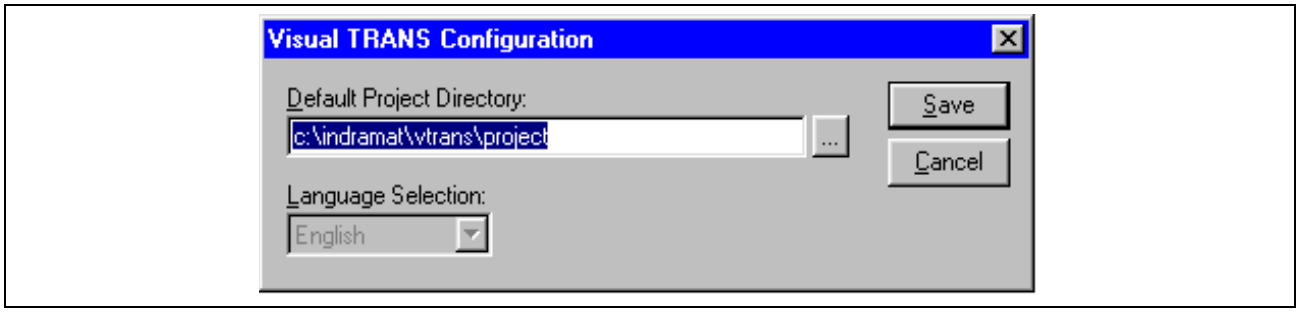


Fig. 1-31: Configuration of Visual TRANS

Configuration allows the user to select the drive and directory that Visual TRANS information is located. Select the "***...***" button to change the drive using the mouse, or click in the white area beneath "*Default Project Directory:*" and enter the complete path from the keyboard.

Once the desired path is entered, click on ***Save*** to retain the new Visual TRANS directory or ***Cancel*** to return to the previously active Visual TRANS application.

"*Language Selection:*" is currently under development and is not available for this release of Visual TRANS. Visual TRANS is currently available in English only.

DeviceNet...

Used to configure the card parameters used by the DCF01.x Fieldbus card. These parameters are:

Parameter	Description
C-0-2630	DeviceNet ID (Drop Number) Valid values 0 to 63_{10} (0 to $3F_{HEX}$)
C-0-2633	DeviceNet Baud Rate Valid values: • 125000 Baud ($1E848_{HEX}$) • 250000 Baud ($3D090_{HEX}$) • 500000 Baud ($7A120_{HEX}$)
C-0-2636	Fieldbus Word/Byte Order Swapping Valid values: • 0 (Neither swapped) • 1 (Word order swapped) • 2 (Byte order swapped) • 3 (Both Word and Byte order swapped)

The **DeviceNet...** function is only valid if Visual TRANS is connected on-line with a TRANS 01-D with Version 07VRS firmware. If not on-line, or if on-line with a control with older version firmware, opening **DeviceNet...** from the **Setup** menu will produce the message displayed in Fig. 1-32.

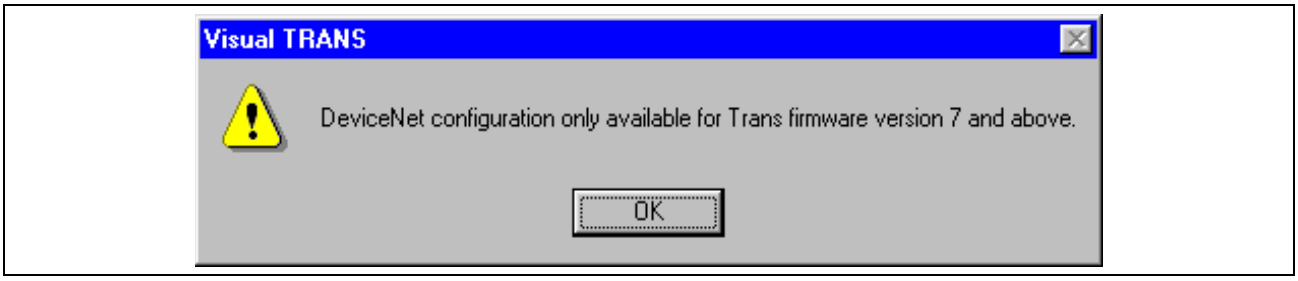


Fig. 1-32: DeviceNet Warning Window

The card parameters exist and can be set in a Version 07VRS control regardless of whether the DCF01.x card is installed or which I/O type is chosen in **Drive I/O...** configuration. The parameters can be changed in any mode of the TRANS 01-D.

The currently configured *Drop Number* is displayed in a box(see Fig. 1-33). It can be changed by clicking on the arrow for the pull-down menu, scrolling to the desired number, and clicking on that number.

The currently configured *Baud Rate* is displayed in a box (see Fig. 1-33). It can be changed by clicking on the arrow for the pull-down menu and then clicking on the desired choice.

Byte Order and *Word Order* are chosen to be swapped or not swapped by clicking the corresponding radio buttons. The currently selected choice is indicated by the dot.

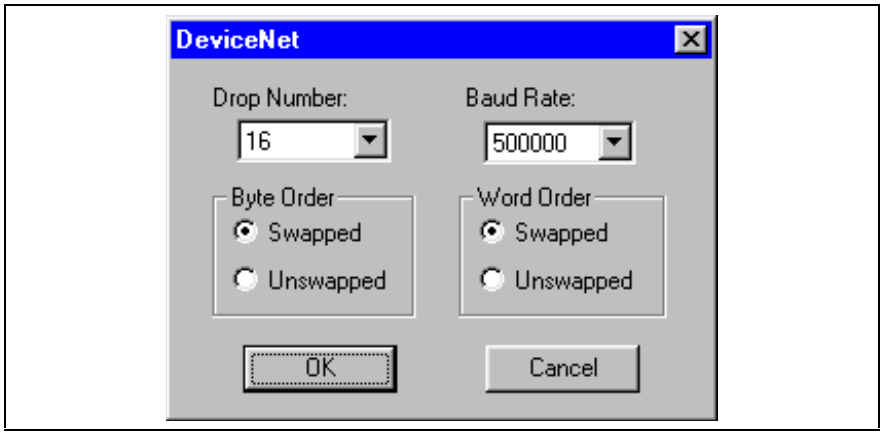


Fig. 1-33: DeviceNet Window

When all settings are made, clicking ***OK*** sends all of the parameters to the card and the **DeviceNet** window is closed. If the **DeviceNet** window is closed with ***Cancel*** or the ***Close*** button in the upper right corner, the card parameters are left unchanged.

Drive I/O...

Used to specify the drive input/output (I/O) configuration for off-line programming. Upon selecting **Drive I/O...** the following screen will appear:

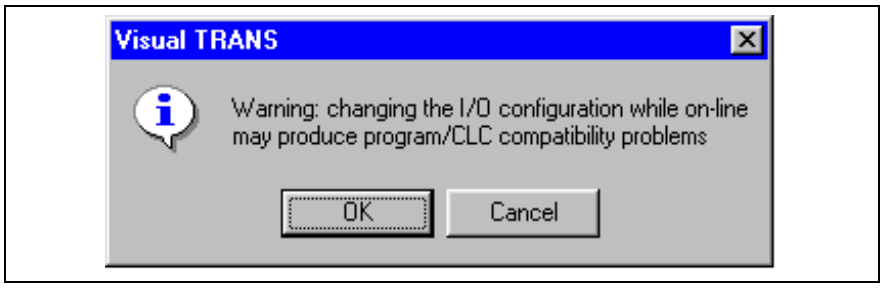

Visual TRANS
Warning: changing the I/O configuration while on-line may produce program/CLC compatibility problems
OK
Cancel

Fig. 1-34: Drive I/O Warning Screen

Select ***OK*** to continue, or ***Cancel*** to exit, the Drive I/O configuration application.

After ***OK*** is selected, the following screen will appear.

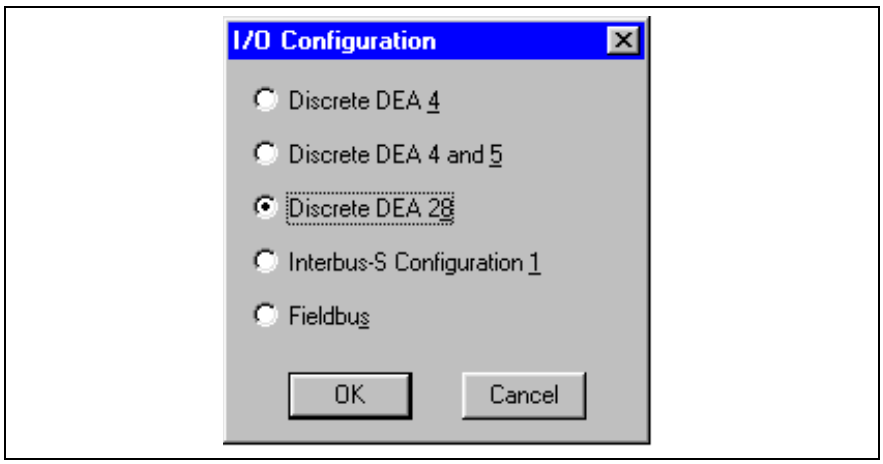


Fig. 1-35: Drive I/O... Screen

Select either Discrete DEA 4, Discrete DEA 4 and 5, DEA 28, Interbus-S Configuration 1, or Fieldbus and then confirm the choice by clicking ***OK***. (A black dot indicates the selected I/O option.)

Drives...

See ***Chapter 4*** for detailed information on the ***CLC Drive Parameter Editor*** (containing the **Oscilloscope** function).

Drives Help Directories…

Used to set the path(s) to the drive parameter help file(s). The Parameters Overview screen (see ***Chapter 4***, ***CLC Drive Parameter Editor***) uses this to display context sensitive help for a specific drive parameter. If you do not have the correct help files for your drive, they can be requested from an Indramat office.

The left edit field is used to enter the first 13 characters of the drive firmware type code. This value can be read from the ***CLC Drive Parameter Editor*** screen (see ***Chapter 4***).

The right edit field is used to enter the complete path of the drive help file containing the parameter descriptions. The last character in the filename is 'P' or 'p'. Clicking on the "***...***" button will allow easy selection of the path to the drive parameter help file.

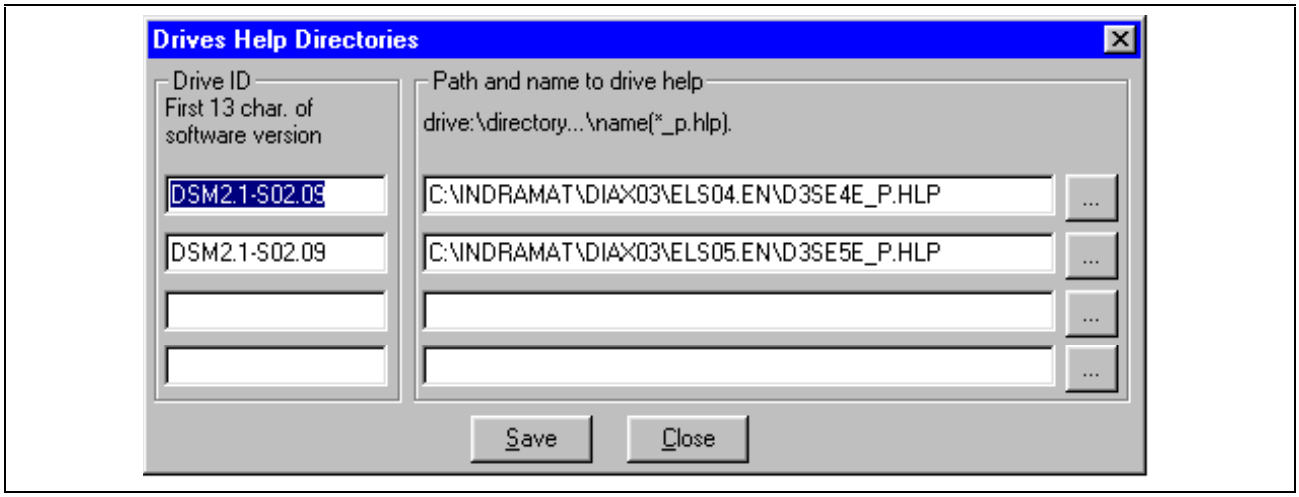


Fig. 1-36: Drive Help Directories Screen

Tools Menu

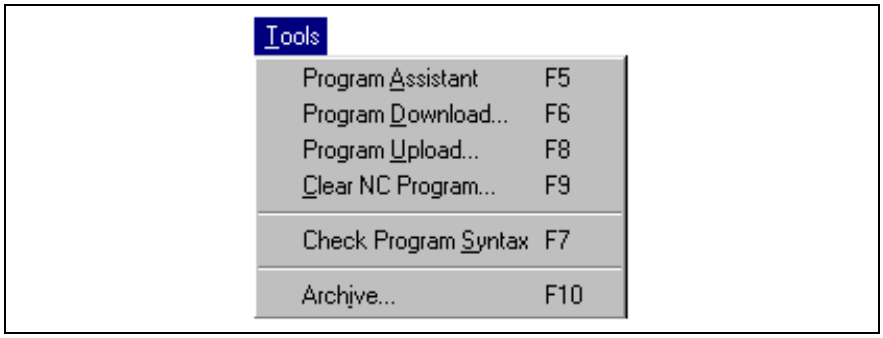


Fig. 1-37: Tools Drop-Down Menu

Program Assistant... F5

This function provides a step-by-step method that aids with the creation of a TRANS 01-D program. See "***Creating a Program Block***" for more information.

Program Download... F6

Used to send NC program information from the PC based Visual TRANS to the TRANS 01-D. This option is only available when the TRANS 01-D and the PC are connected and on-line. When selected, the following screen will appear.

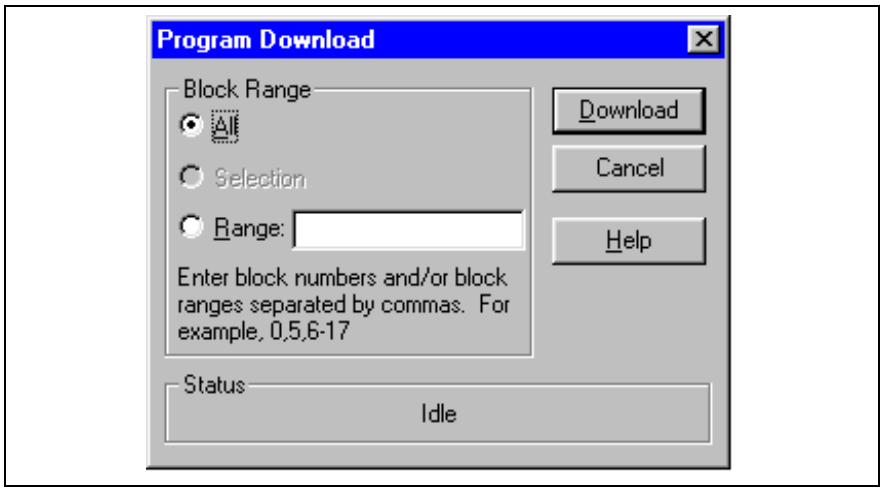


Fig. 1-38: Program Download Screen

Select the *Block Range* and click on ***Download*** to execute or ***Cancel*** to exit the **Program Download… F6** command.

Program Upload... F8

Used to retrieve NC program information from the TRANS 01-D and copy it to Visual TRANS on a PC. (This option is only available when the TRANS 01-D and the PC are connected and on-line.) When selected, the following screen will appear.

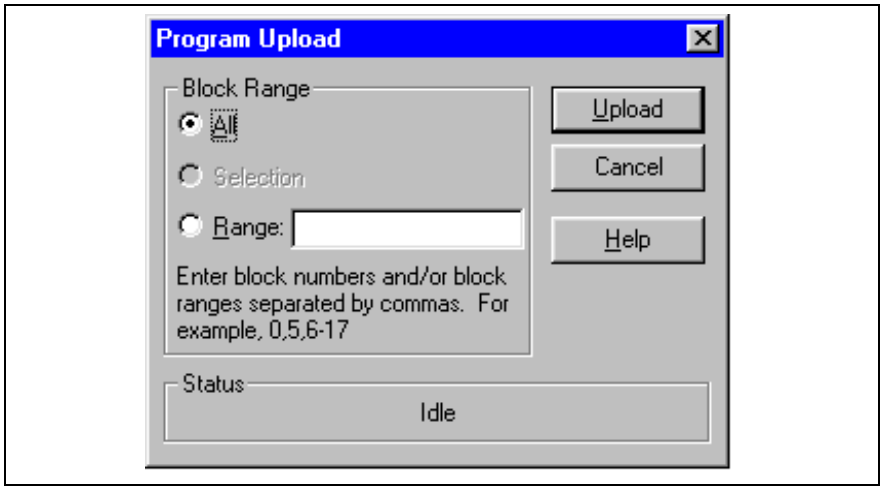


Fig. 1-39: Program Upload Screen

Select the *Block Range* and click on ***Upload*** to execute or ***Cancel*** to exit the **Program Upload… F8** command.

Clear NC Program… F9

Deletes the current NC program in the TRANS 01-D.

Once cleared, the TRANS 01-D program cannot be recovered.

When **Clear NC Program… F9** is selected, the following screen appears.

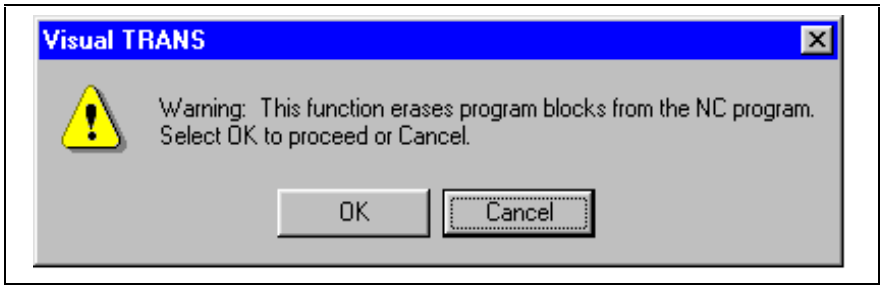


Fig. 1-40: Clear NC Program… F9 (First Screen)

To quit this application, select ***Cancel***, or to continue this application, select ***OK*** to reveal the following screen.

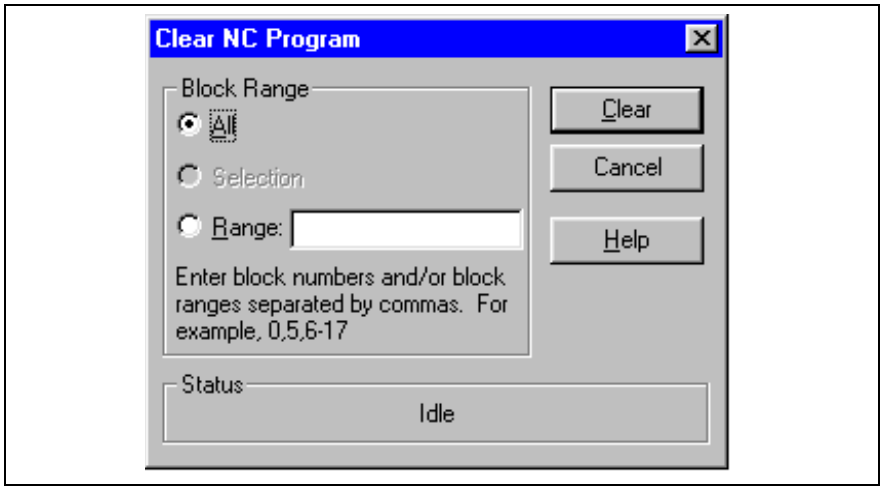


Fig. 1-41: Clear NC Program… F9 (Second Screen)

Enter the *Block Range* and select ***Clear*** to execute or ***Cancel*** to exit the **Clear NC Program** function.

Check Program Syntax F7

Used to check an NC Program Block or NC part program for syntax errors to verify that input coding is properly formatted with valid codes that will be accepted by the TRANS 01-D.

Check for syntax errors in the NC program by pressing ***F7*** on the keyboard or clicking on "**Check Program Syntax F7**". If the status bar shows "No errors found in program," the NC program is error-free. If errors are detected, the NC program block will be highlighted and the status bar will display an error message with a tip on the error detected. If the NC program was previously saved, the corrected version of the NC program should be saved to replace the erroneous NC program.

Archive… F10

The **Archive… F10** function is used to *Backup* and *Restore* parameters (*System* and *Drive*), *NC Programs*, and *Tool Correction* values. This function is also used to create the *Upload Parameters as Text* and *CTA 10-1/BTC06 Parameter* reports.

The following warning window is displayed when the **Archive… F10** function is executed and the system is not in “Parameter” mode.

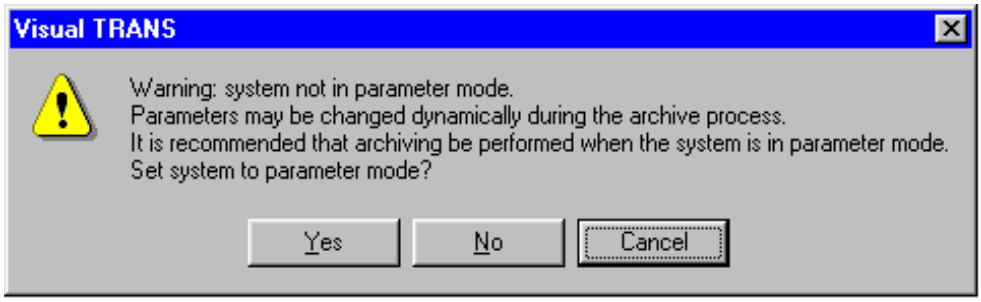

Visual TRANS
Warning: system not in parameter mode.
Parameters may be changed dynamically during the archive process.
It is recommended that archiving be performed when the system is in parameter mode.
Set system to parameter mode?
Yes
No
Cancel

Fig. 1-42: Not In Parameter Mode Warning Window

If ***Yes*** is clicked on, then the system is automatically placed in “Parameter” mode and the **Archive System** window is opened (see below).

If ***No*** is clicked on, then the system is **not** placed in “Parameter” mode and the **Archive System** window is opened.

If ***Cancel*** is clicked on, then the **Archive… F10** function is exited.

The following four warning windows are independently displayed, respectively, when the **Archive… F10** function is executed and either (1) the system is in “Automatic” mode; (2) an “NC program” is active (i.e., running in “Manual” mode); (3) the system is in “Programming” mode; or (4) the “Host” has been disabled.

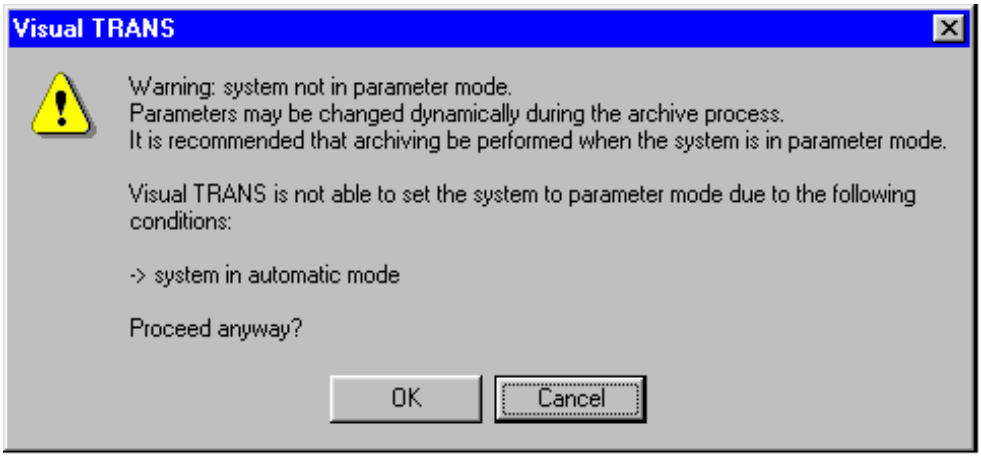

Visual TRANS
Warning: system not in parameter mode.
Parameters may be changed dynamically during the archive process.
It is recommended that archiving be performed when the system is in parameter mode.
Visual TRANS is not able to set the system to parameter mode due to the following conditions:
-> system in automatic mode
Proceed anyway?
OK
Cancel

Fig. 1-43: In Automatic Mode Warning Window

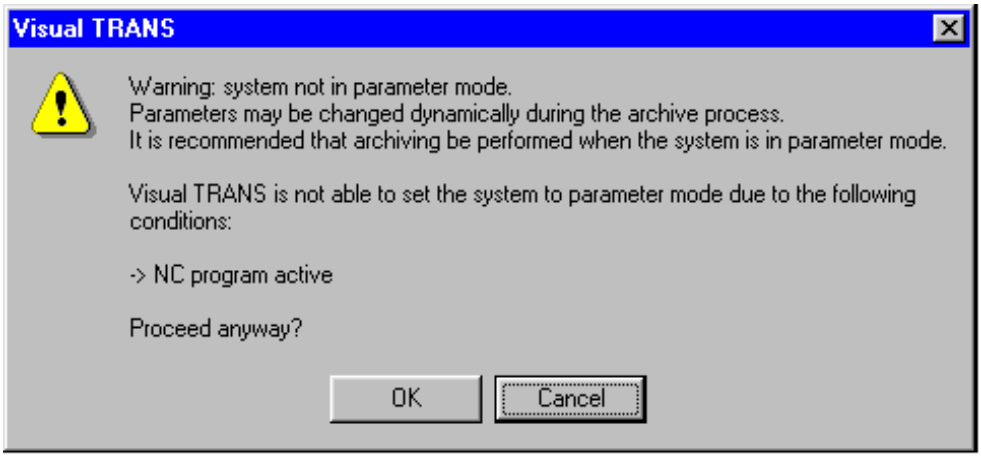

Visual TRANS
Warning: system not in parameter mode.
Parameters may be changed dynamically during the archive process.
It is recommended that archiving be performed when the system is in parameter mode.
Visual TRANS is not able to set the system to parameter mode due to the following conditions:
-> NC program active
Proceed anyway?
OK
Cancel

Fig. 1-44: NC Program Is Active Warning Window

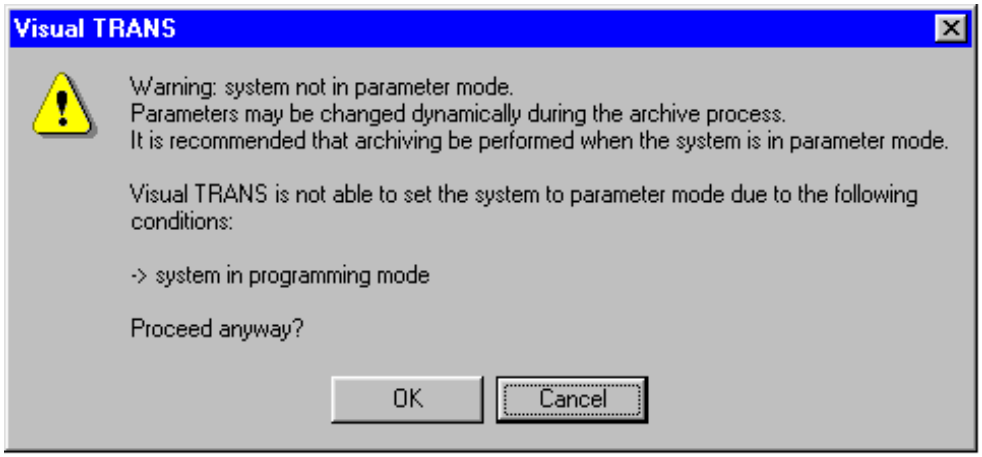

Visual TRANS
Warning: system not in parameter mode.
Parameters may be changed dynamically during the archive process.
It is recommended that archiving be performed when the system is in parameter mode.
Visual TRANS is not able to set the system to parameter mode due to the following conditions:
-> system in programming mode
Proceed anyway?
OK
Cancel

Fig. 1-45: In Programming Mode Warning Window

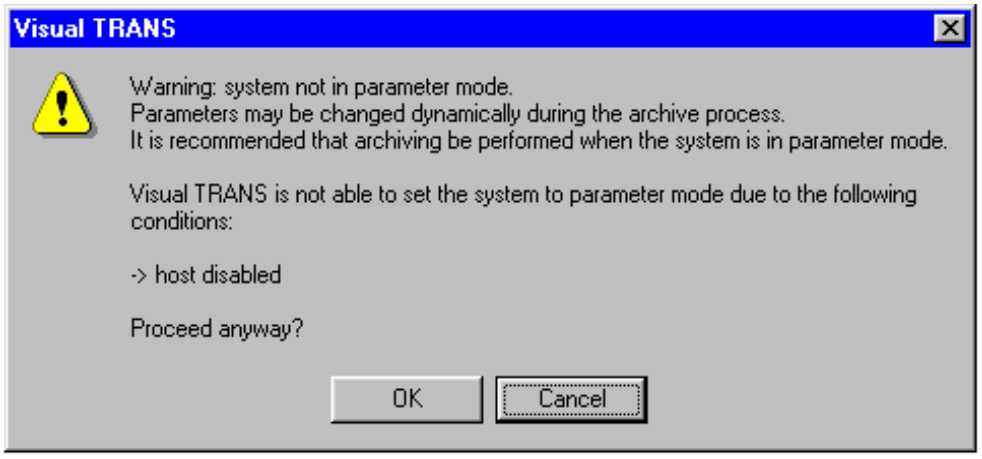

Visual TRANS
Warning: system not in parameter mode.
Parameters may be changed dynamically during the archive process.
It is recommended that archiving be performed when the system is in parameter mode.
Visual TRANS is not able to set the system to parameter mode due to the following conditions:
-> host disabled
Proceed anyway?
OK
Cancel

Fig. 1-46: Host Disabled Warning Window

If ***OK*** is clicked on, then the **Archive System** window is opened (see below).

Note: In these scenarios, the system will **not** be in “Parameter” mode.

If ***Cancel*** is clicked on, then the **Archive… F10** function is exited.

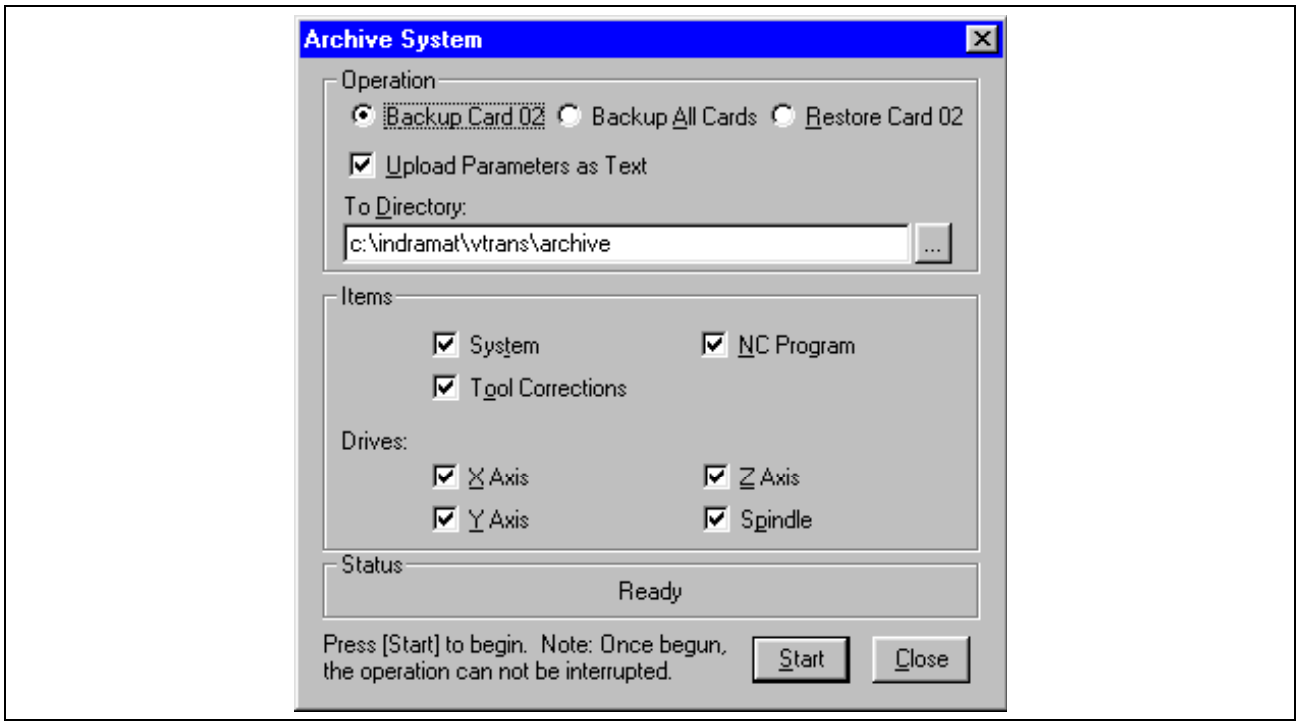


Fig. 1-47: Archive - Backup

Backup Card

When the **Archive System** window is initially opened, the *Backup* option is chosen and all of the check boxes are checked. Furthermore, the *To Directory* defaults to *c:\indramat\vtrans\archive*. The user may change this default directory path by clicking on the "***...***" button. The **Browse Directories** window is opened when the "***...***" button is clicked on. At this point, the user may either open up a different existing directory path or create a new directory path by clicking on the ***Create...*** button.

Note: Any *Drives* that do not exist on the system will be grayed out.

When the ***Start*** button is clicked on:

- A subdirectory will be created under the path defined in *To Directory* named ***Card##_yymmdd***, where *##* is the current card number and *yymmdd* is the numeric date.
- The *System* files will be backed up from the CLC-D card and stored in the *c:\indramat\vtrans\archive**Card##_yymmdd\System.prm*** file.
- The *Upload Parameters as Text* report will create the *c:\indramat\vtrans\archive**Card##_yymmdd\System.txt*** file.
- The *Tool Corrections* will be backed up from the CLC-D card and stored in the *c:\indramat\vtrans\archive**Card##_yymmdd\T_corr.prm*** file.
- The *X Axis* data [if configured] will be backed up from the respective drive and then this data will be stored in the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive01.prm*** file.
- The *Upload Parameters as Text* report will create the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive01.txt*** file for the *X Axis* [if configured].

- The *Y Axis* data [if configured] will be backed up from the respective drive and then this data will be stored in the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive02.prm*** file.
- The *Upload Parameters as Text* report will create the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive02.txt*** file for the *Y Axis* [if configured].
- The *Z Axis* data [if configured] will be backed up from the respective drive and then this data will be stored in the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive03.prm*** file.
- The *Upload Parameters as Text* report will create the *c:\indramat\vtrans\archive**lCard##_yymmdd\Drive03.txt*** file for the *Z Axis* [if configured].
- The *Spindle* data [if configured] will be backed up from the respective drive and then this data will be stored in the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive04.prm*** file.
- The *Upload Parameters as Text* report will create the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive04.txt*** file for the *Spindle* [if configured] .

Note 1: The "**.txt*" files that are created when the *Upload Parameters as Text* report is initially run (i.e., the check box for *Upload Parameters as Text* is checked) can only be created/updated if the corresponding check boxes are checked in the *Items* section for *System*, *X Axis*, *Y Axis*, *Z Axis*, and/or *Spindle*.

Note 2: The "**.txt*" files contain user friendly lists of parameters (numbers, names, values, and units) that are sorted by: (1) parameter number prefix in the given order (i.e., "01 S-0-" & "01 P-0-" for the *Drives* and "01 C-0-" & "01 T-0-" / "02 T-0-" / "03 T-0-" / "04 T-0-" & "01 A-0-" / "02 A-0-" / "03 A-0-" / "04 A-0-" for the *System*) and (2) the actual 4 digit parameter number.

- The *NC Program* will be backed up from the CLC-D card and stored in the *c:\indramat\vtrans\archive**Card##_yymmdd\Nc_prog.pgm*** file.
- The *CTA 10-1/BTC06 Parameter* report automatically runs in the background when the *System* check box and at least one of the 4 *Drives* check boxes are checked. The *CTA 10-1/BTC06 Parameter* report creates the following file:
  - *c:\indramat\vtrans\archive**Card##_yymmdd\Parametr.txt***. This file contains a listing of *Process* and *Axis* parameters (with values and units) for all configured axes as they appear on either the CTA 10-1 or the BTC06.

The progress of the *Backup* can be monitored with the use of the message strings that are displayed in the *Status* section of the **Archive System** window.

Any of the check boxes that default to checked can be manually unchecked by clicking on the appropriate check box before the ***Start*** button is clicked on.

When the ***Close*** button is clicked on after a *Backup* has successfully completed, the **Archive System** window will close and the system will automatically come out of "Parameter" mode [assuming that the system was originally placed in "Parameter" mode].

Backup All Cards

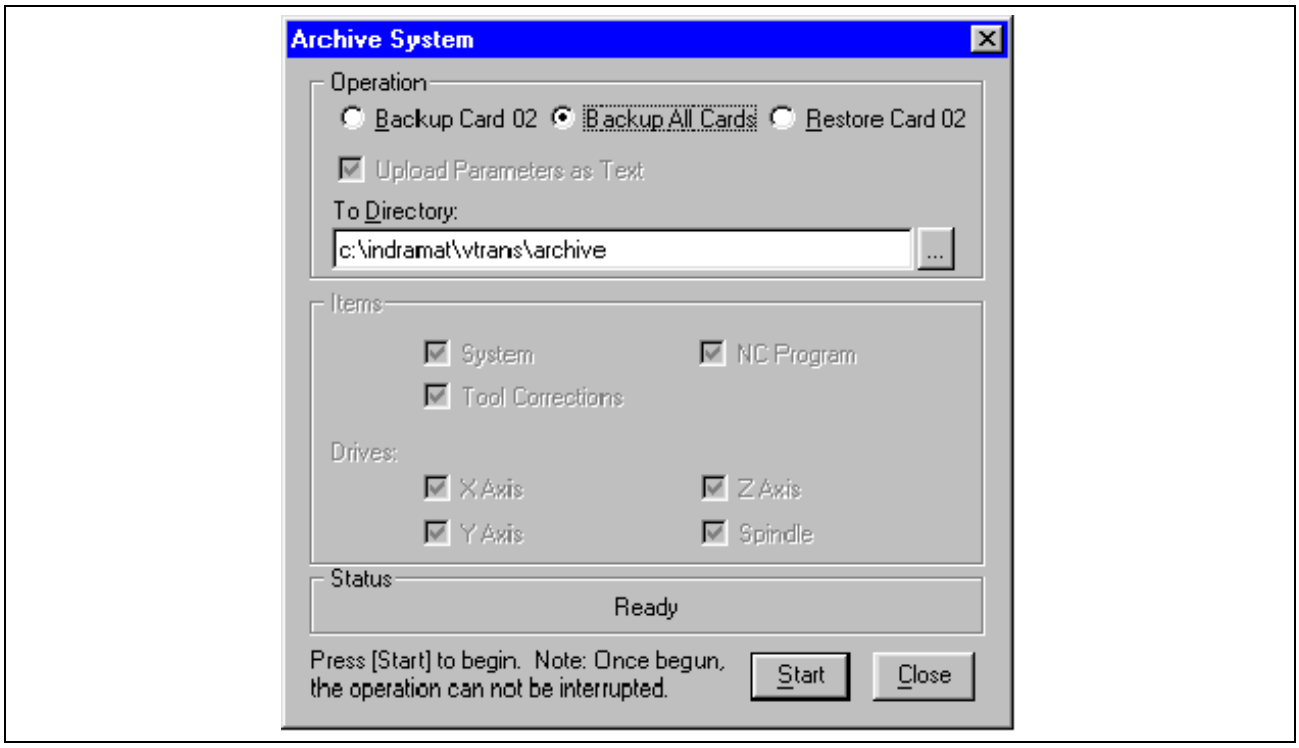


Fig. 1-48: Archive - Backup All Cards

The *Backup All Cards* function provides a way to backup a number of cards on an RS-485 network with a single command. The polling function similar to that in the **Setup** Menu **Card Selection...** searches all addresses to determine which card numbers are connected (see Fig. 1-49). All connected cards or some group of them can be selected for backup. All available data for the connected cards will be backed up. It is not possible to individually deselect any items. *System*, *Tool Corrections*, *NC Program*, and all available *Drives* will be backed up. *Upload Parameters as Text* will also be done as well as the creation of the *Parametr.txt* report.

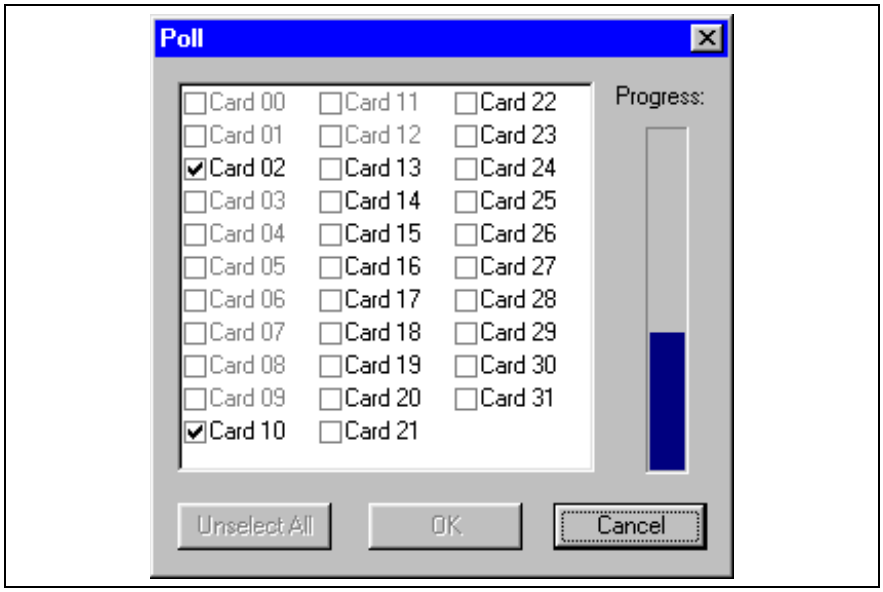


Fig. 1-49: Archive – Backup All Cards - Poll Window

Before choosing, *Backup All Cards,* choose the desired directory for the backup in the *To Directory* box. As soon as *Backup All Cards* is selected, polling begins. The card addresses that do not respond are grayed and those that do respond are checked so that by default all found cards will be backed up. The progress is indicated by the rising progress bar at the right of the window. When all addresses have been polled, it is possible to individually uncheck or check any of the available cards. It is also possible to ***Unselect All*** with the button and then select the desired cards by clicking the check box in front of the desired cards.

Pressing ***OK*** displays a new window (see Fig. 1-50) with checkmarks in front of those cards that are selected for backup.

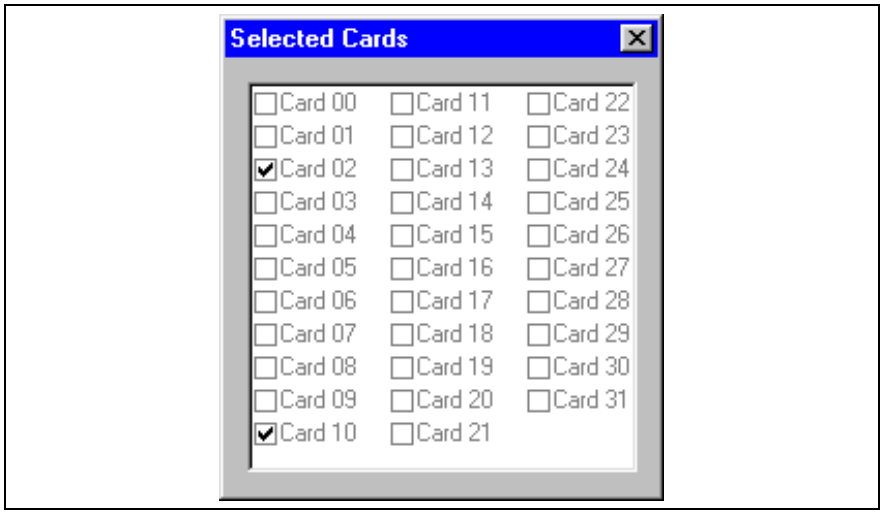


Fig. 1-50: Archive – Backup All Cards – Selected Cards Window

The *To Directory* box is still available to confirm that the desired target directory has been chosen. It can still be changed if desired.

All other checkboxes in the **Archive System** window are grayed and cannot be individually selected or deselected.

Pressing ***Start*** begins the *Backup All Cards* process. The current activity and progress is indicated in the *Status* box. For each selected card, the backup proceeds as described in *Backup Card ##* above. The same subdirectory structure, report log, and files are created.

The only differences between using *Backup All Cards* and individually selecting each card and doing *Backup Card ##* are:

- Once started, the *Backup All Cards* operates unattended until all cards are done.
- If a card responds when the initial polling is done but is unavailable when its turn comes for the backup, its backup will be attempted one more time after all selected cards are done. If it does not respond on the second attempt, it will be logged as failed.
- A single **Backup*yymmdd*.log** file is created in the subdirectory chosen in the *To Directory* window. It shows the backup status of each card address, whether it was found, whether it was backed up and to what subdirectory. It also has a summary of the number of cards successfully backed up and the number that failed.
- *Backup All Cards* does not attempt to change the TRANS 01-D to “Parameter” mode during the backup. It does the backup in whatever mode the TRANS 01-D is at the time.

After all cards are done, the *Status* indicates “Ready”. Clicking on ***Close*** will close the **Archive System** and **Selected Cards** windows.

In a network environment where all of the TRANS 01-D CLC-D cards on a machine are connected via RS-485 to a computer running Visual TRANS

and each of those computers is networked to a server via Ethernet, it is possible to back up all data to a central location. To automate this process, it is necessary to know the path of To Directory where Visual TRANS is storing the log files. By checking the date used in the names of the log files, it is possible to determine the date of the latest available backup. Then by uploading and opening that log file, the complete path to each card backup can be read. To facilitate the parsing of the information within the log file, the information is delimited into fields by commas. When opened or imported as a comma delimited file to a program like Excel, the fields (or columns) are:

1. Action or target;
2. Result or status;
3. Date;
4. Time;
5. Subdirectory name.

Using the root path of the "Archive Directory" in the first record of the log file and then all subdirectories in the fifth column of the log file, it is possible to locate all data that must be copied to the central backup server.

Restore Card

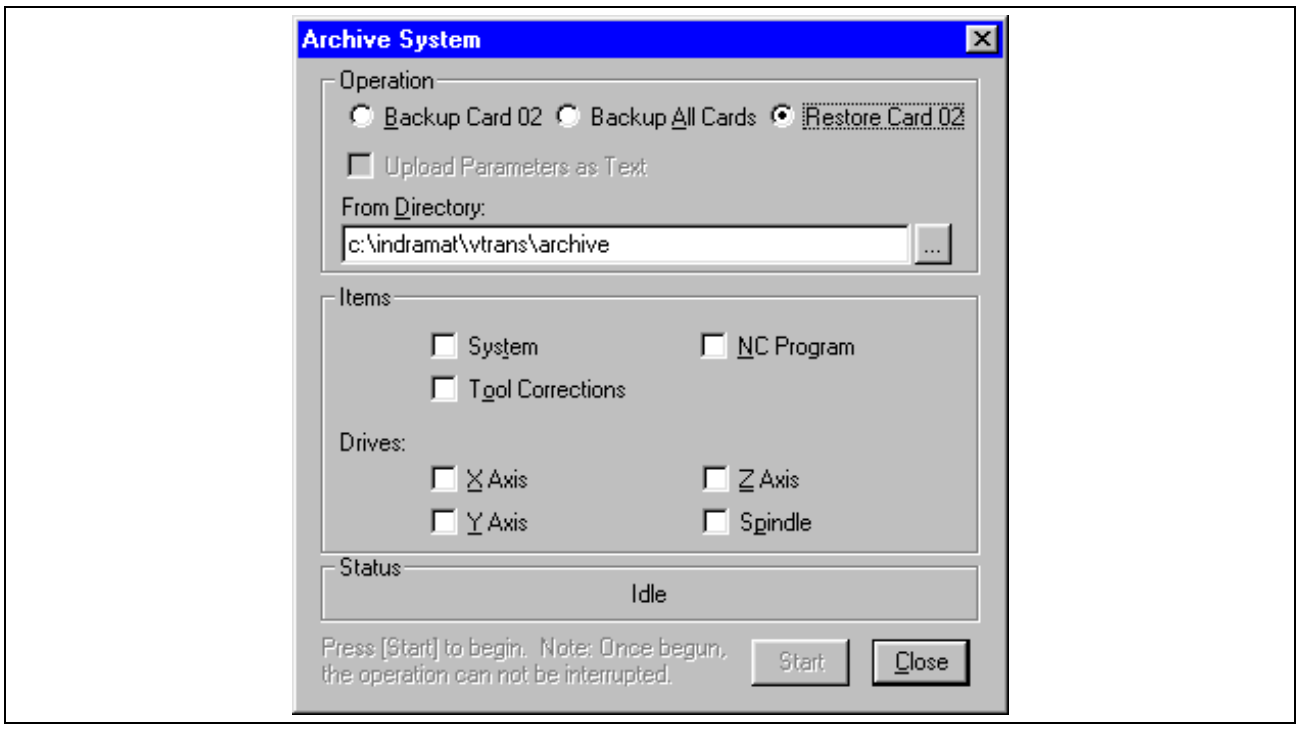


Fig. 1-51: Archive - Restore

When the *Restore* option is chosen, all of the check boxes are unchecked and the ***Start*** button is grayed out. Furthermore, the *Upload Parameters as Text* field has been grayed out.

Note: The *Upload Parameters as Text* and *CTA 10-1/BTC06 Parameter* reports are disabled when the *Restore* option is chosen.

The *To Directory* defaults to *c:\indramat\vtrans\archive*. The user may change this default directory path by clicking on the "***...***" button. The **Browse Directories** window is opened when the "***...***" button is clicked on. At this point, the user may either open up a different existing directory path or create a new directory path by clicking on the ***Create...*** button.

Note: The user may wish to change the *To Directory* to the following subdirectory that was created for the given card when the respective *Backup* was performed (if applicable): *c:\indramat\vtrans\archive**\Card##_yymmdd*** [where *##* is the current card number and *yymmdd* is the numeric date].

The following field(s) will be independently grayed out in the *Items* section when the corresponding files are missing:

- The *System* field will be grayed out when the *c:\indramat\vtrans\archive**Card##_yymmdd\System.prm*** file does not exist.
- The *Tool Corrections* field will be grayed out when the *c:\indramat\vtrans\archive**Card##_yymmdd\T_corr.prm*** file does not exist.
- The *X Axis* field [if configured] will be grayed out when the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive01.prm*** file does not exist.
- The *Y Axis* field [if configured] will be grayed out when the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive02.prm*** file does not exist.
- The *Z Axis* field [if configured] will be grayed out when the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive03.prm*** file does not exist.
- The *Spindle* field [if configured] will be grayed out when the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive04.prm*** file does not exist.
- The *NC Program* field will be grayed out when the *c:\indramat\vtrans\archive**Card##_yymmdd\Nc_prog.pgm*** file does not exist.

The user should place a check mark in each check box for all existing *Items* that are to be *Restored*. The ***Start*** button will automatically be enabled when the first check mark is placed in the respective check box in the *Items* section.

When the ***Start*** button is clicked on:

- The *System* files [if selected] will be restored from the *c:\indramat\vtrans\archive**Card##_yymmdd\System.prm*** file to the CLC-D card.
- The *Tool Corrections* [if selected] will be restored from the *c:\indramat\vtrans\archive**Card##_yymmdd\T_corr.prm*** file to the CLC-D card.
- The *X Axis* data [if selected] will be restored from the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive01.prm*** file to the respective drive.
- The *Y Axis* data [if selected] will be restored from the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive02.prm*** file to the respective drive.
- The *Z Axis* data [if selected] will be restored from the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive03.prm*** file to the respective drive.
- The *Spindle* data [if selected] will be restored from the *c:\indramat\vtrans\archive**Card##_yymmdd\Drive04.prm*** file to the respective drive.
- The *NC Program* [if selected] will be restored from the *c:\indramat\vtrans\archive**Card##_yymmdd\Nc_prog.pgm*** file to the CLC-D card.

Note 1: If the *NC Program* field has been selected for the *Restore*, then a **Visual TRANS** window will be displayed after the ***Start*** button is clicked on. This **Visual TRANS** window reads: "Clear existing NC program blocks prior to restore?" Two buttons exist on this window: ***Yes*** and ***No***. Click on the ***Yes*** button if you wish to delete the entire existing NC program from the CLC-D card. Click on the ***No*** button if you wish to *Restore* the previously backed up NC program (i.e., *c:\indramat\vtrans\archive**Card##_yymmdd****Nc_prog.pgm***) over the program that currently exists on the CLC-D card. The user should keep in mind that if an NC program exists on the CLC-D card, then only the NC blocks that exist in the ***Nc_prog.pgm*** file will be downloaded to the CLC-D card during the *Restore*. When this occurs, it is possible that NC blocks from the old program could be merged in between NC blocks in the *Restore*d NC program, and vice versa. This could create problems with respect to the flow of the NC program.

Note 2: The system is automatically placed in "Programming" mode when the ***Nc_prog.pgm*** file is restored to the CLC-D card.

The progress of the *Restore* can be monitored with the use of the message strings that are displayed in the *Status* section of the **Archive System** window.

When the ***Close*** button is clicked on after a *Restore* has successfully completed, the **Archive System** window will close and the system will automatically come out of "Parameter" mode [assuming that the system was originally placed in "Parameter" mode].

Data Menu

When selecting the **Data** option, the following drop-down menu will appear.

Fig. 1-52: Data Drop-Down Menu

Registers

This function allows viewing and editing of registers on the CLC-D card. A maximum number of 512 registers exist on the CLC-D card.

Choosing the **Registers** menu item from the **Data** menu opens a dialog box (see Fig. 1-53) listing the register ID numbers, the symbolic label names associated with the registers (from the *C:\…\Vtrans\Register.lst* file), and the corresponding data values for the registers. When the dialog box opens, it automatically uploads and displays the current contents of the registers.

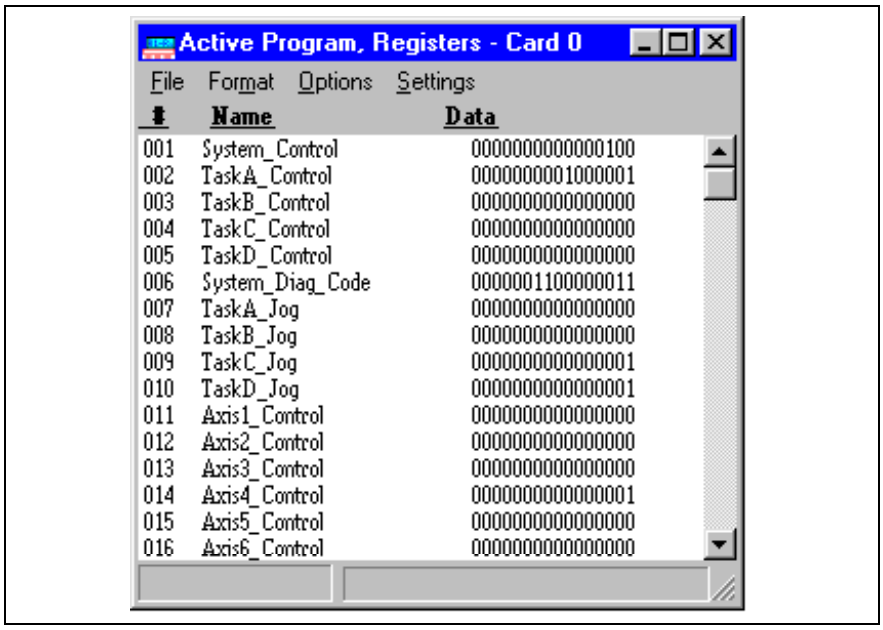


Fig. 1-53: Main Registers Screen

The **Format** pull-down menu allows you to change the displayed format of the register data. The available display formats are *Hex*, *Binary*, and *Decimal*.

The **Options** pull-down menu shows the available F-key commands for editing (**Edit Bits F2** and **Edit Register F3**), forcing (**Force a Register F4**), and clearing (**Clear All Forcing F5**). Selecting an I/O register list entry and pressing ***F3*** opens a dialog box (see Fig. 1-54) which allows the value in the selected I/O register table to be changed. Clicking the ***Save*** button downloads the changed value to the CLC-D card.

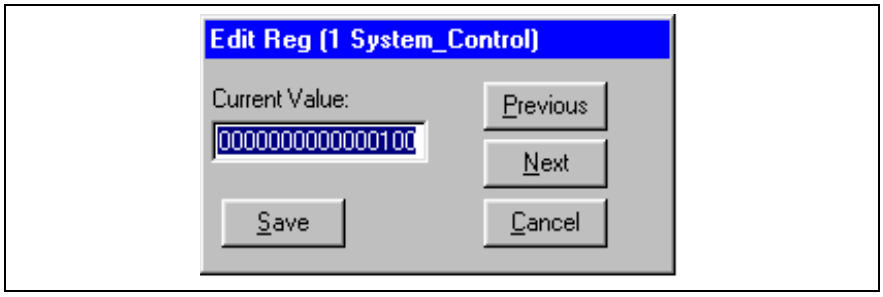


Fig. 1-54: Edit Register Screen

Selecting an I/O register list entry and pressing ***F2***, or simply double clicking on an I/O register list entry, opens a dialog box (see Fig. 1-55) for the selected register and displays the bit names and values associated with the register. Bit names are obtained from the " *C:*\...*Vtrans**Bit.lst* file.

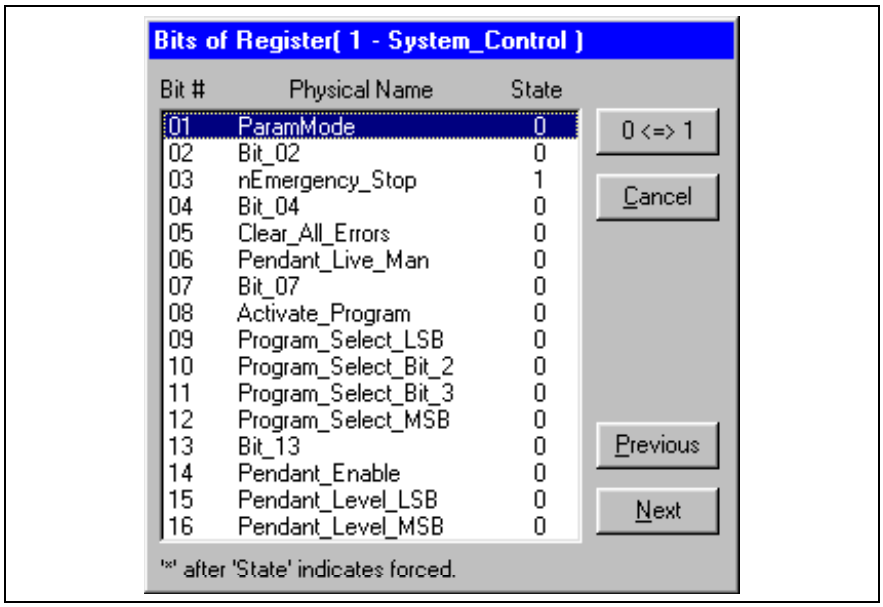


Fig. 1-55: Edit Register Bit Screen

Selecting a register bit list item and clicking on the "***0 <=> 1***" button inverts the current state of the bit and downloads the change of state to the CLC-D card through the serial communication link. The next periodic upload from the CLC-D card reflects the change of state in the displayed list.

Bits controlled by the CLC-D card I/O, user program, or I/O Mapper will not be altered using the "***0 <=> 1***" toggle button. However, their state may be forced using the **Force a Register F4** option (see Fig. 1-56). Items denoted with an "**x**" in the rightmost column have forcing in effect for the associated bit.

System installation and troubleshooting may require directly changing the state of register bits without depending on the I/O sub-system. Selecting a register and pressing F4 opens a Register Forcing dialog box that allows you to setup a forced bit during system setup and debugging.

Fig. 1-56: Register Forcing Screen

Forced bits cannot be affected by an NC program or the I/O sub-system. Forcing directly accesses the CLC-D system's I/O lines. Bits that are forced will remain in the forced state until the forcing is changed, the CLC-D card is reset, or power is cycled on-to-off.

Forcing employs a forcing mask and a forcing state of 16-bit control words allowing you to change a single bit, or combination of bits, within the register without affecting the other bits.

The forcing mask value enables which bits may be affected by forcing. The forcing state value determines the actual state of the enabled bits. The following algorithm mathematically describes how the final register state is set:

new register state = (old register state & inverted forcing mask) | (forcing state & forcing mask)

Example:

forcing mask	0000 0000 0000 1000
forcing state	0000 0000 0000 1000
inverted forcing mask	1111 1111 1111 0111
old register state	0000 1101 1010 0001
new register state	0000 1101 1010 1001

All forcing can be cleared with the use of the **Clear All Forcing F5** function.

Selecting the **Settings** function opens the following window:

Fig. 1-57: Card Selection Setup Screen

The **Settings** function is identical to the **Card Selection...** function which resides in the **Setup Menu**.

WARNING	**Bit forcing directly changes the state of the CLC's inputs and outputs. Forcing I/O bits can result in harm to people and equipment. Make sure you fully understand all the effects on the system that could result from forcing an I/O line.**

Status Menu

Diagnostic Log

This function lists the last 100 errors the CLC-D card has encountered. The date, time, and extended error codes are displayed along with the error messages.

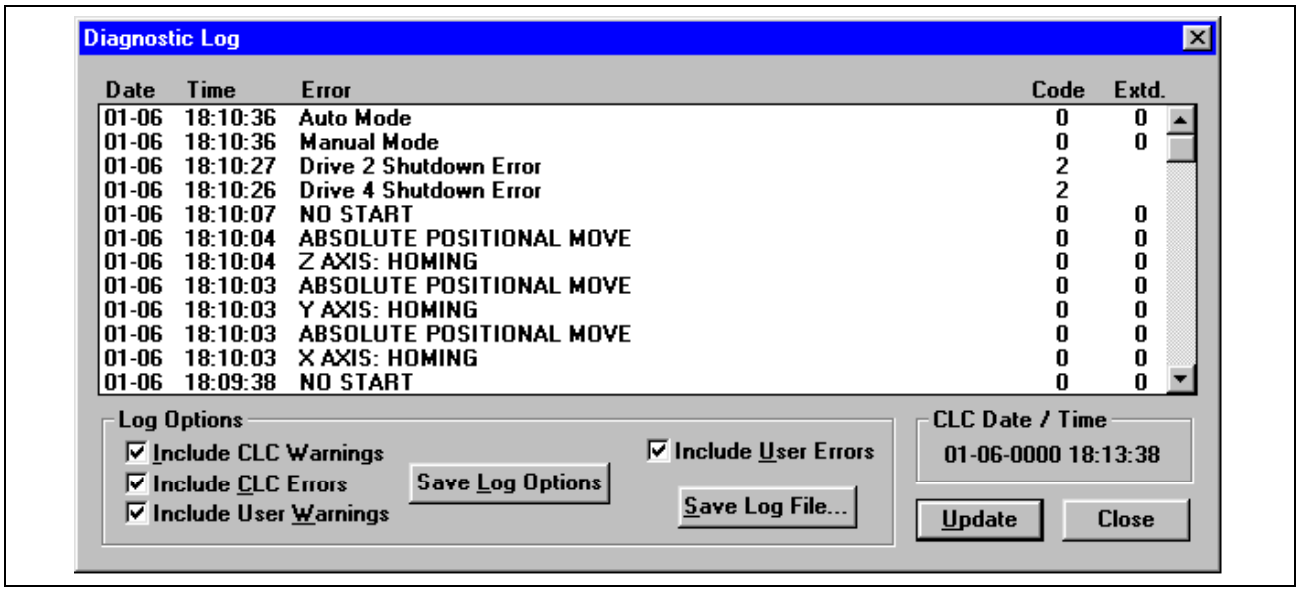

Diagnostic Log
Date Time Error Code Extd.
01-06 18:10:36 Auto Mode 0 0
01-06 18:10:36 Manual Mode 0 0
01-06 18:10:27 Drive 2 Shutdown Error 2
01-06 18:10:26 Drive 4 Shutdown Error 2
01-06 18:10:07 NO START 0 0
01-06 18:10:04 ABSOLUTE POSITIONAL MOVE 0 0
01-06 18:10:04 Z AXIS: HOMING 0 0
01-06 18:10:03 ABSOLUTE POSITIONAL MOVE 0 0
01-06 18:10:03 Y AXIS: HOMING 0 0
01-06 18:10:03 ABSOLUTE POSITIONAL MOVE 0 0
01-06 18:10:03 X AXIS: HOMING 0 0
01-06 18:09:38 NO START 0 0
Log Options
Include CLC Warnings
Include CLC Errors
Include User Warnings
Save Log Options
Include User Errors
Save Log File...
CLC Date / Time
01-06-0000 18:13:38
Update
Close

Fig. 1-58: Diagnostic Log Screen

Note: The date and time are relative to the power on of the CLC-D card, there is no battery backed clock.

The **Save Log Options** button allows the user to customize which of the 4 log options will be displayed when this function is executed. The user has the ability to check/uncheck any combination of the 4 check boxes associated with the log options and save these settings for future use.

The **Save Log File…** button will save the diagnostic log that is currently being displayed to a file named "*Diaglog.txt*". The "*Diaglog.txt*" file will be stored in the default subdirectory of *C:\…\Vtrans\Project*. The user has the ability to override both the file name and the storage location.

The **Update** button will upload the diagnostic log information that exists on the CLC-D card at the time that the button is clicked on.

Modes

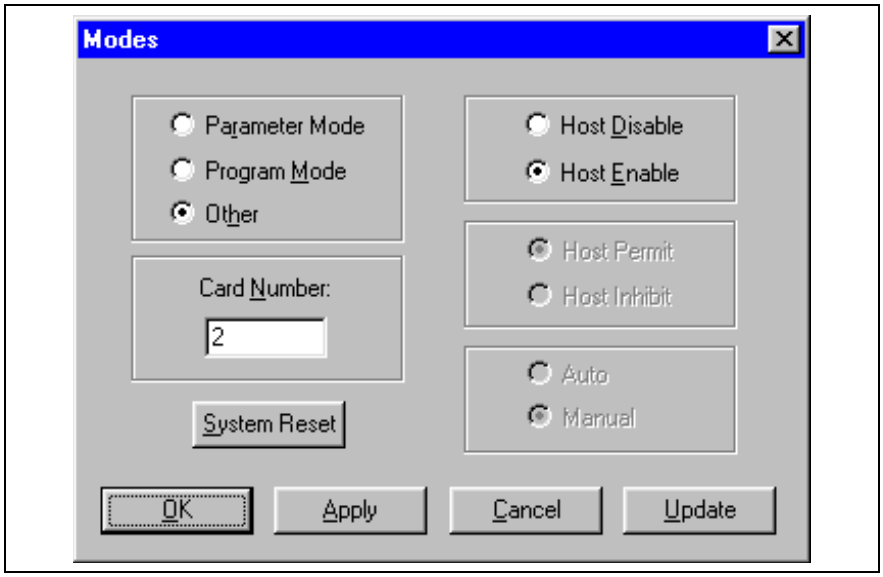


Fig. 1-59: Modes Window

Choosing **Modes** from the **Status** menu opens the **Modes** window. This provides status display and remote control of signals that are important for the archiving and program editing functions of Visual TRANS. With this screen, it should be possible to do all required functions for replacing and restoring program and parameters to a TRANS 01-D without the use of a CTA / BTC HMI.

The **Modes** window displays the status of:

- *Parameter Mode* – Register 106, bit 1
- *Program Mode* – Register 106, bit 2
- *Card Number* – Card Parameter C-0-0002 (Unit Number)
- *Host Disable* / *Enable* – Register 106, bit 7
- *Host Permit* / *Inhibit* – Register 106, bit 8
- *Auto* / *Manual* – Register 106, bit 16

Auto / *Manual* and *Host Permit* / *Inhibit* are signals that are controlled only by inputs from the line controller (PLC fieldbus or discrete I/O) so they are for status display only.

Card Number is the "Unit Number" parameter stored on the TRANS 01-D to which Visual TRANS is currently communicating. A change in this parameter does not become active until the TRANS 01-D card is rebooted (by cycling power off and on or by doing a ***System Reset***). If the parameter has been changed but not yet activated by rebooting, the *Card Number* shown on this screen may be different from the card address chosen in **Card Selection** from the **Setup** menu, that determines which card number the Visual TRANS program sends as the address in its messages. This is especially important when using Visual TRANS to restore parameters to a new blank card because the *Card Number* parameter will be defaulted to 0 and must be changed to the proper number for that station and rebooted to activate before the archived parameters can be restored.

Parameter Mode, Programming Mode, and *Host Disable* / *Enable* are the same status and request flags that are used by the CTA / BTC HMI. If Visual TRANS determines that the proper conditions exist to change the state of these flags, they are shown in normal black lettering. If an interlock prevents changing a flag, then that signal name is grayed. For example if in *Auto* mode, it is not possible to request *Programming Mode*, *Parameter Mode*, or change *Host Disable* / *Enable* so they are all grayed. If the name is not grayed, it can be selected by clicking on the name of the desired mode or the radio button circle in front of it. It can also be selected by typing the shortcut key ("r" for *Parameter Mode*, "m" for *Program Mode*, "h" for *Other*, "d" for *Host Disable*, "e" for *Host Enable*, "p" for *Host Permit*, "i" for *Host Inhibit*). When the selections are set as desired, they are sent by clicking the ***Apply*** button or typing the shortcut letter "a".

Note that the graying is based on whether the TRANS 01-D was in *Auto* mode when the screen was last updated. When the ***Apply*** is done, another check is done to confirm that it is not in *Auto* mode at that moment. If it has changed and it is in *Auto* mode, the request is not sent and the updated states will be shown in the **Modes** window. The *Programming Mode* and *Parameter mode* controls will then be grayed.

Host Disable / *Host Enable* is ungrayed only if the following conditions are met:

- Not in "Auto" mode – Register 106 bit 16 = 0
- "Program" is not paused – Register 106 bit 10 = 0
- "Program Active" is low – Register 106 bit 9 = 0
- "Hand " mode is low – Register 106 bit 3 = 0
- Not in "Program" mode – Register 106 bit 2 = 0

- Not in “Parameter” mode – Register 106 bit 1 = 0
- “Manual Spindle Enable” is low – Register 100 bit 10 = 0

To read the current status of the information from the TRANS 01-D and refresh the display, click the ***Update*** button or press the shortcut letter “u”.

To leave the **Modes** window without sending any selections that might have been changed, click the ***Cancel*** button or press the shortcut letter “c”.

Click ***OK*** or press the shortcut letter “o” to send the selected information the same as Apply and then close the window the same as Cancel.

The other function provided in the **Modes** window is a button for ***System Reset***. The System Reset button is grayed if a reset cannot be performed for any of the following reasons:

- The TRANS 01-D is not in either Manual or Automatic mode. (E.g. in Programming or Parameter mode)
- The Run output is on (i.e., NC program execution is running – not stopped or paused)

When the ***System Reset*** button is not grayed, clicking on that button or pressing the shortcut letter “s” brings up a warning window:

Caution: ***System Reset*** restores all modes and internal data equivalent to turning the power off and on (see Fig. 1-60).

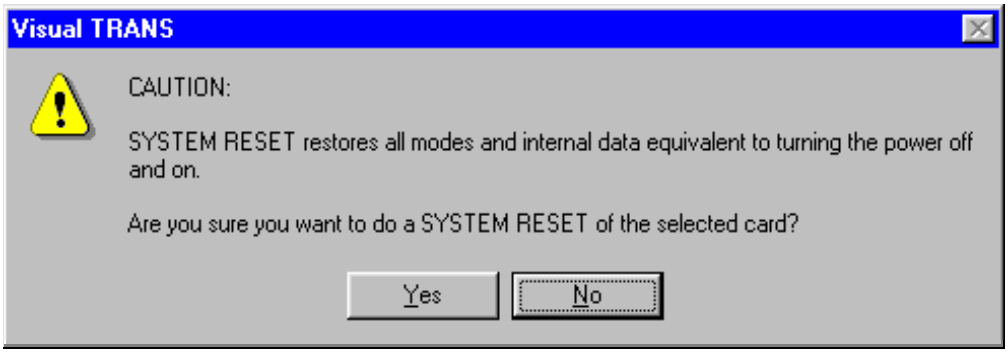

Visual TRANS
CAUTION:
SYSTEM RESET restores all modes and internal data equivalent to turning the power off and on.
Are you sure you want to do a SYSTEM RESET of the selected card?
Yes
No

Fig. 1-60: Modes – System Reset Warning Window

Pressing ***No*** or typing the shortcut “n” cancels and returns to the Modes window without sending the System Reset command. Pressing ***Yes*** or typing the shortcut letter “y”, sets Program Reset and then Fault Clear requests high simultaneously. This causes the card to reboot in the same way as it does when requested from the line controller (PLC fieldbus or discrete I/O). While the TRANS 01-D card is rebooting, communication with Visual TRANS will stop. It is not possible to update the status of the modes until the restart is completed.

System

Choosing **System** from the **Status** menu opens the **System** tab window. Tabs also exist for **Task A**, **Task B**, **Task C**, and **Task D**.

The **System** tab displays information about the current CLC-D card hardware and software for the indicated unit number. Information is also displayed for language selection, operating mode, SERCOS phase, and diagnostic messages.

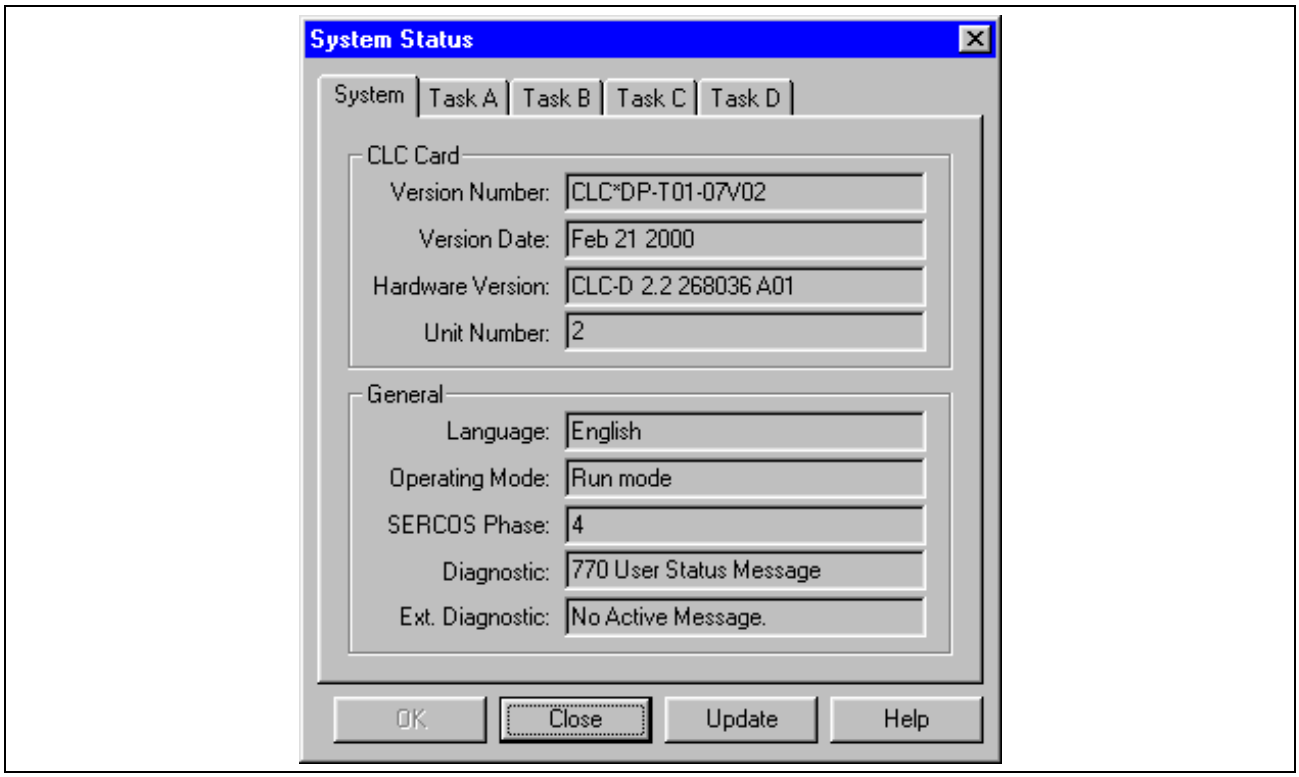


Fig. 1-61: System Status Screen – System Tab

Selecting the **Task A**, **Task B**, **Task C**, or **Task D** tab from the **System Status** window uploads data regarding the current status of the selected CLC-D task.

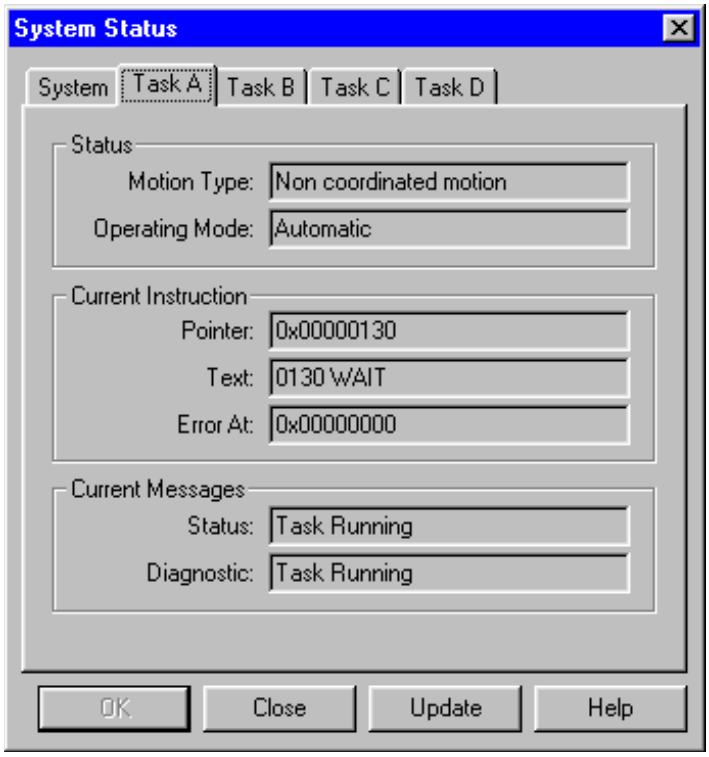


Fig. 1-62: System Status Screen – Task A Tab

The *Status* section indicates the type of motion programmed in the selected task for the active NC program and the current CLC-D mode (i.e., Parameter Mode, Task Running, Manual Mode, etc.)

The *Current Instruction* section displays the NC program instruction that is executing, the pointer to this NC program instruction, and a pointer to a run-time error (if applicable).

The *Current Messages* section displays the status and diagnostic messages.

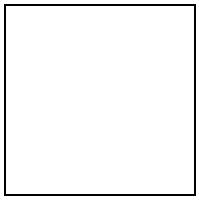
1.6 Tool Bar

Located below the Menu Bar, the Tool Bar contains the following icons.

Fig. 1-63: Visual TRANS Tool Bar

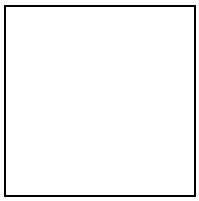
Those icons, which control functions unique to the Visual TRANS program, are described in this section.

Program Assistant



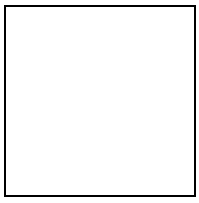
This function provides a step-by-step method that aids with the creation of a TRANS 01-D program. See "***Creating a Program Block***" for more information.

Program Download…



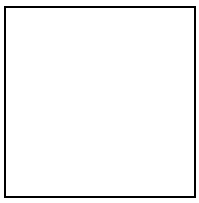
Used to send NC program information from the PC based Visual TRANS to the TRANS 01-D. See "***Program Download... F6***" for more information.

Program Upload…



Used to retrieve NC program information from the TRANS 01-D and copy it to Visual TRANS on a PC. See "***Program Upload... F8***" for more information.

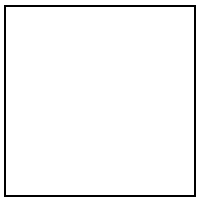
Clear NC Program…



Deletes the current NC program in the TRANS 01-D. See "***Clear NC Program… F9***" for more information.

Once cleared, the TRANS 01-D program cannot be recovered.

Check Program Syntax

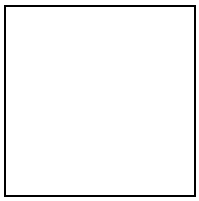


Used to check an NC program block or an NC part program for syntax errors to verify that input coding is properly formatted with valid codes that will be accepted by the TRANS 01-D. See "***Check Program Syntax F7***" for more information.

Archive…

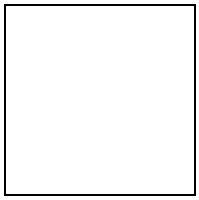
Use this command to archive CLC parameters. See "***Archive… F10***" for more information.

About Visual TRANS…



Use this command to display the copyright notice, the version number of your copy of Visual TRANS, and other pertinent information.

Help



Use the Context Help command to obtain help on some portion of Visual TRANS. When you choose the Toolbar's Context Help button, the mouse pointer will change to an arrow and question mark. Then click somewhere in the Visual TRANS window, such as another Toolbar button. The Help topic will be shown for the item you clicked. This tool bar button is currently under development.

2 CLC DDE SERVER

2.1 Dynamic Data Exchange

The Microsoft® Windows™ operating system specifies a method for transferring data between applications. This is called **Dynamic Data Exchange (DDE)**. DDE is a message protocol that developers can use for exchanging data between Microsoft® Windows™-based applications. The CLC communication server uses the **Dynamic Data Exchange Management Library (DDEML)** which is built on top of the DDE protocol. The DDEML provides services that the message-based DDE protocol does not support. Under the DDEML a client application requests information from a server application, or it sends unsolicited data to the server. The client does this by passing predefined ASCII strings to the server through the DDEML.

Before a client and server can exchange data, they must first agree upon what they are going talk about. This is done by establishing a conversation. Conversations are defined by a service name and a topic name. The CLC server application uses this information to specify how and who to communicate with. After having established a conversation, the client application can now pass data. This is done by specifying an item name. The item name identifies the specific data to be passed.

There are three basic types of data transactions that can be initiated by the client application. A *request* transaction is used to obtain data from the server. The server application knows how to obtain the requested information. The second type of transaction is an *advise link*. After a client application establishes an advise link with a server, it is up to the server to poll the data for changes. If the server finds that the data has changed it will notify the client application. The third type of transaction is a *poke*. A poke transaction is used to send data for a specific item to the server.

The Dynamic Data Exchange Server

The CLC DDE is a Microsoft® Windows™ based **Dynamic Data Exchange (DDE)** server application that is used to communicate with Indramat's CLC motion control cards. It has been implemented using the Microsoft® Windows™ **Dynamic Data Exchange Management Library (DDEML)**.

Dynamic Data Exchange Interface

A windows application, known as a *client,* can pass information between other applications known as *servers* using **Dynamic Data Exchange (*DDE*)**. A client establishes a conversation with a server specifying a *Service* and a *Topic*. Once a conversation has been started, a client may request or send information by specifying an *item*.

Service Name

The CLC communication server supports two DDE service names. The standard service name is ***CLC_DDE***. This should be used for all connections except when connecting to a CLC compiled program file. For this case ***CLC_FILE*** should be used.

Topic Name

When the standard service name is used to exchange CLC data, the topic name identifies the method of connection to the CLC card and the card unit number. Valid strings consist of a communication device name and a

unit number. Valid device names are ***SERIAL_***, ***AT_MODEM_***, ***XYCOM_***, ***GE_P&P_***, ***DEMO_*** or ***ISA_*** and valid card unit numbers are '0' to 'F'. Connections that use the CLC_FILE service should specify the CLC program file as the topic name. If the file is not located in the same directory as clc_dde.exe then the complete path should be included. To exchange server data the service name should be CLC_DDE and the topic name should be **SERVER**. This is the only topic which will not support an advise link. *See section SERVER Topic Name**.***

Example: **"SERIAL_0"** Serial connection to a CLC card designated as unit '0'.

"SERVER" Exchange CLC_DDE server information.

Item Name

The item name identifies the specific data to exchange. When exchanging CLC data the item name consists of a string that contains the class, subclass and data identifiers of the information for the CLC card. The strings follow the ASCII serial protocol. When exchanging server data the item name should consist of the section and entry name from the INI file (clc_dde.ini). The two names must be divided by a pipe ('|') character. Not all server data has read/write capabilities.

Example: **"RX 0.10"** Specifies register 10 in hexadecimal format.

"TP 2.20" Specifies task B parameter 20.

"CP 1.122" Specifies card parameter 122.

2.2 The Communication Servers Main Window

The **CLC DDE Server** window displays the unit number and current status for the selected CLC control card. To display the status for a different CLC card or to disable this feature, open the server configuration dialog box under the settings menu item. Select the desired connection/unit from the CLC status display combo box.

When the CLC DDE is in an icon state the tip of the arrow will change colors depending on the communication state. A green tip means that the server is actively communicating, and a red tip indicates that the server is in an error state. If the monitored CLC card's status indicates an error state while the application is an icon, the server window will be restored to the normal state.

When the **CLC DDE Server** is running, either the icon or the dialog box below is displayed.

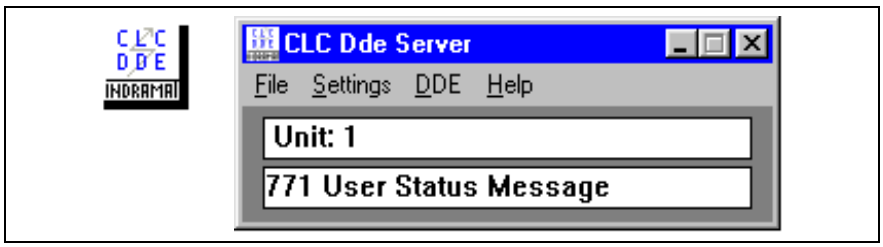


Fig. 2-1: CLC DDE Server – Main Screen

If the icon is displayed, double-clicking the icon restores the dialog box. The **CLC DDE Server** dialog box contains four selections on the main menu bar: **File**, **Settings**, **DDE**, and **Help**. The **Settings** and **DDE** selections will be discussed below.

Settings Menu - CLC Server Configuration

The **Server Configuration** function allows setting of various system parameters as well as providing performance status information.

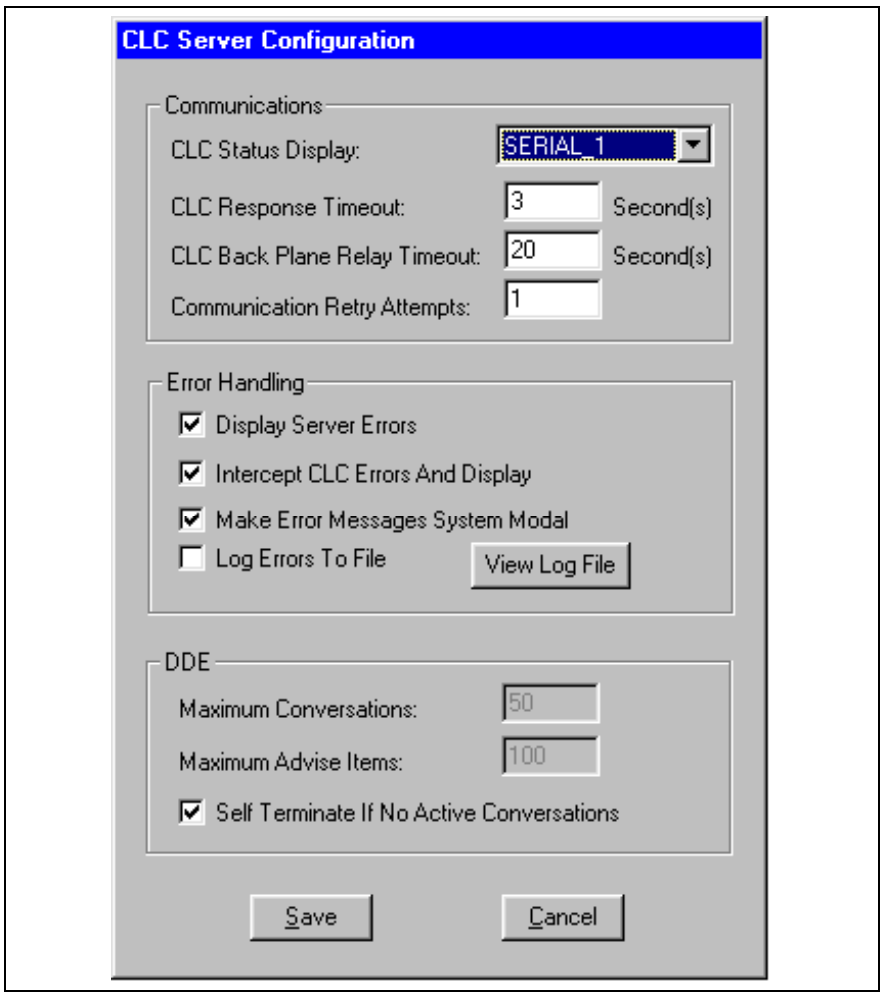


Fig. 2-2: Configuration Screen

Communications

CLC Status Display	Selects the CLC device/unit (i.e. serial_0) combination to be displayed in the status window of the server. The request will be inserted into the standard client advise loop queue. Selecting “Disable Status” can turn off this feature.
CLC Response Time-out	The amount of time in seconds that the server will wait for a completed response from the CLC control card before diagnosing a disconnect. The valid range of values is 1-900 seconds.
CLC Back Plane Relay Time-out	CLC-V control cards have the ability to redirect incoming serial messages over the VME back plane to other CLC-V cards in the same rack. This allows a host to address multiple control cards with one serial connection. These transmissions may require more time than a direct serial link. The relay time-out value is used for these transactions. The valid range of values is 1-900 seconds.
Communication Retry Attempts	The number of times the server will re-send a message before it issues an error. The valid range of values is 0-255.

Error Handling

Intercept CLC Errors And Display	Checking this box will cause the server to intercept CLC error responses and displayed them in a message box. Request and poke transactions will return failure to the client application. Advise links will remain active, however they will return nothing until the error is resolved. The error response will be written to the error log file if that feature is enabled. If this box is not checked the error string will be returned to the client.
Make Error Messages System Modal	Checking this box will cause all server generated message boxes to have system modal attributes. This means that all applications will be suspended until the user responds to the message box. The window can not be forced to the background.
Log Errors To File	Checking this box will cause the server to log all server errors to a file. The current system date and time will be associated with each log entry. As a default this feature is not enabled.
View Log File	Pressing this button will cause the current error log file to be displayed in notepad.

DDE

Maximum Conversations	This is a static display of the maximum number of allowed DDE conversations as specified in the INI file. The server will refuse any DDE connection requests in excess of this value.
Maximum Advise Items	This is a static display of the maximum number of allowed DDE advise links as specified in the INI file. The server will refuse any requests for advise links in excess of this value.
Self Terminate If No Active Conversations	Checking this box will cause the server to close itself when the last DDE conversation with it has terminated. This is the default state.

Settings Menu - Serial Communications

The **Serial Communications** function allows the user to select the serial communication parameters the server will use. When this dialog box is open all communications are suspended. If changes are made to the configuration they will take effect when the ***Save*** button is pressed.

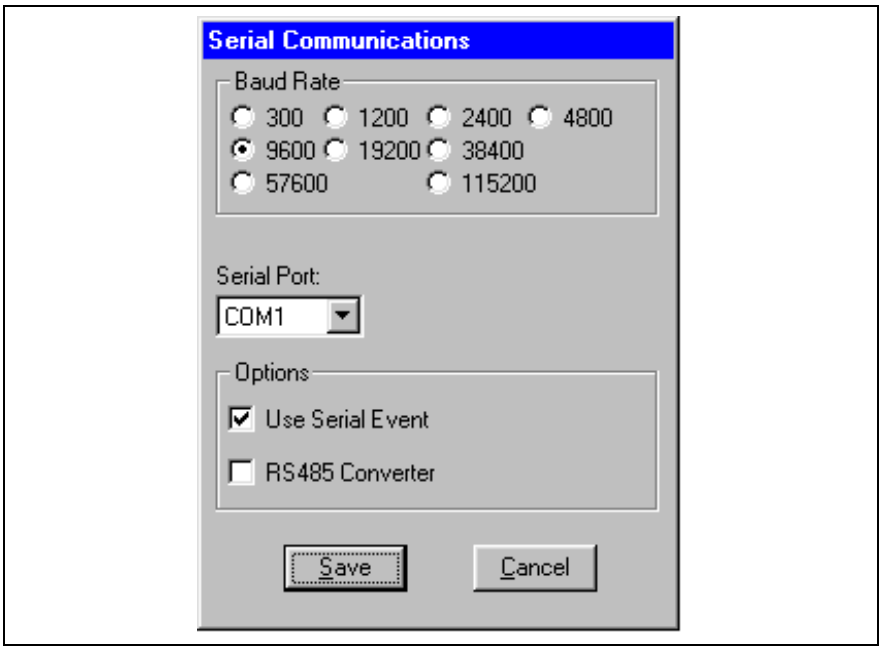


Fig. 2-3: Serial Communications Screen

Baud Rate	Check the proper baud rate to use when communicating serially with a CLC card.
Serial Port	Select the serial communications port to use on the PC.
Use Serial Event	Checking this box causes Windows to notify the server when a completed message is in the receive queue. This will increase the number of serial messages sent over polling for a response. Slower computers may not be able to utilize this feature.
RS485 Converter (not available with TRANS 01-D)	This option should be used when an RS232 to RS485 converter is present. A delay will be inserted between messages, which is equal to at least one character transmission at the selected baud rate. This is necessary to ensure that the CLC card has had sufficient time in which to turn the RS485 transmitter off and enable the receiver. Please note that the converter must toggle the transmitter and receiver automatically, and also that echo back must be disabled.

DDE Menu - DDE Conversations

The **DDE Conversation List** dialog box displays the *Conversation*, *Service* and *Topic Handles* for all of the current DDE conversations. The *Item Count* column shows the total number of active advise links, request transactions, and poke transactions. Double click on a specific conversation entry in order to view the item transaction list. A second method is to select the conversation and then use the ***Expand*** button. This dialog box is useful when creating client applications that talk to the CLC communications server.

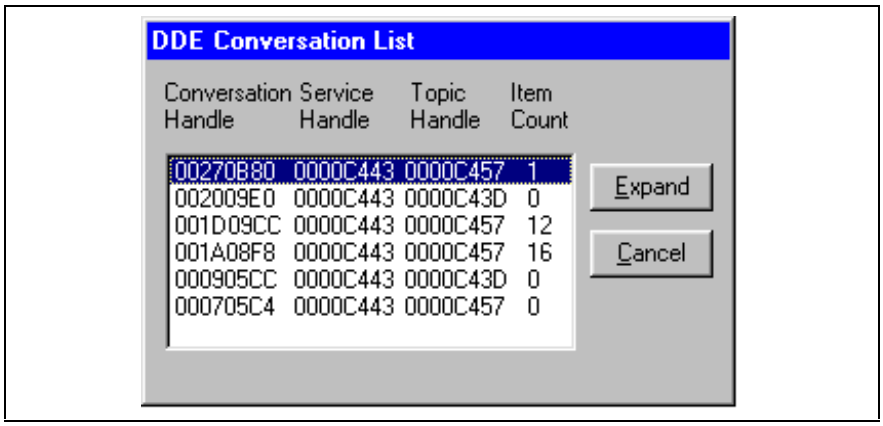


Fig. 2-4: Conversation List Screen

DDE Conversation Item Dialog

The **DDE Conversation Item** dialog box can be used to view the item transaction list for a conversation. The *Service* name, *Topic* string, *Item* string, clipboard *Format* and *Transaction Type* are displayed in text format. Use the ***Next*** and ***Previous*** buttons to cycle through the current list.

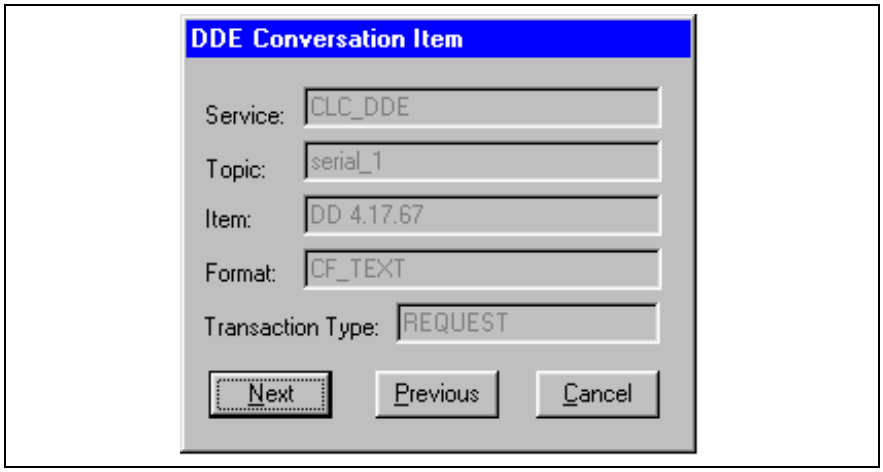


Fig. 2-5: Conversation Item Screen

DDE Menu - Communication Monitor

The DDE **Communication Monitor** function displays all of the current DDE conversations. The **Monitor** window can display DDE requests and/ or responses dependent upon which selection was made under the **Settings** menu.

The active window builds a communications log of all DDE conversations that occur while the monitor is running. Selecting **Clear** will empty the log. Selecting **Stop** will stop the conversation monitoring and allow users to scroll through the log. The **Monitor** window can be resized to enlarge the active viewing area.

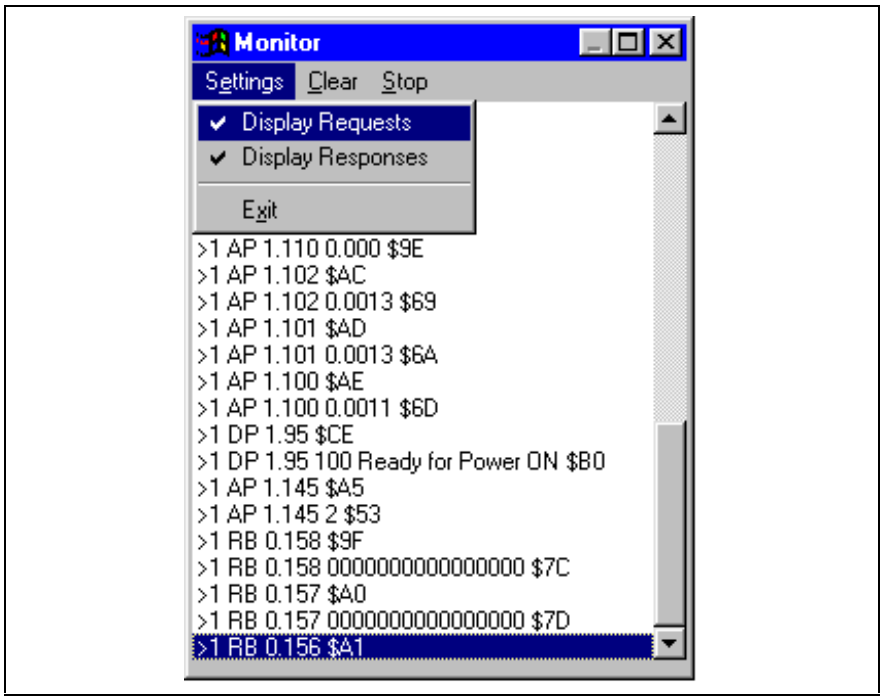


Fig. 2-6: Communication Monitor Screen

2.3 SERVER Topic Name

The “SERVER” topic name allows a DDE client application access to the CLC DDE’s parameter set and status. The server will accept request and poke transactions. When accessing a parameter the client application should specify the section and entry names from the INI file. A pipe character (‘|’) must separate the two names. When requesting status information the client should use “STATUS” as the section name (i.e., “STATUS|ErrorState”). RW = Read/Write and RO = Read Only

Section: GENERAL	Response_Timeout	RW	1-900 Seconds	Message response time out.
	Relay_Timeout	RW	1-900 Seconds	Message time out when using VME pass-through.
	Communication_Retry	RW	0-255	Number of times to re-send a message.
	Suspend_Polling	RO	0 or 1	If 1 CLC_DDE status polling will be disabled.
	Display_CLC_Errors	RW	0 or 1	If 1 CLC_DDE will intercept & display CLC Errors.
	Log_Errors	RW	0 or 1	If 1 all server errors will be logged to the error file.
	Modal_Errors	RW	0 or 1	Displayed errors with the system modal attribute.
	Self_Terminate	RW	0 or 1	Close CLC_DDE when last conversation terminates.
	Monitor_List_Size	RW	1-500	# of entries in communication monitor window.
	Editor	RW	256 Characters	Name & path of text editor to use to display error log.

Section: SERIAL	Baudrate	RO	115200.. 300	Baud rate for serial connection to CLC card.
	Port	RO	1-4	COM port number to use for serial connection.
	Serial_Event	RW	0 or 1	Use serial event option to increase performance.
	RS485_Converter	RW	0 or 1	Activate RS485 adapter code.
Section: VME	Sustain_Bus	RW	0 or 1	Release every cycle option for XYCOM PC.
	A32_Addressing	RW	0 or 1	Use A32 addressing for XYCOM PC.
	VME_IRQ	RO	0-7	Number of VME IRQ to use (0 = disabled).
Section: AT_MODEM	Baudrate	RO	9600..300	Baud rate to use to communicate with the modem.
	Port	RO	1-4	COM port number the modem is on.
	Auto_Connect	RW	0 or 1	Initialize & connect on conversation connection.
	Phone	RW	50 Characters	Phone number to dial.
	Initialize_Script	RW	100 Characters	Script to initialize modem.
	Disconnect_Script	RW	100 Characters	Script to disconnect modem.
	Dial_Prefix	RW	50 Characters	Script to send to modem before phone number.
	Escape_Sequence	RW	50 Characters	Script to send modem to return to command mode.
Section: PC	PC_IRQ	RO	0 or 1	if 1 use PC interrupt for communications.
Section: DDE	Status	RO	200 Characters	CLC_DDE's status request item.
	Max_Conversations	RO	1-3274	Maximum allowed conversations.
	Max_Advise_Items	RO	1-3500	Maximum allowed advise items.
Section: STATUS	ErrorState	RO	0 or 1	If 1 CLC_DDE is issuing an error.
	ErrorText	RO	256 Characters	Error text message CLC_DDE is displaying.
	RequestState	RO	0 or 1	If 1 CLC_DDE is actively communicating.

3 Programming

This chapter contains:

- Application programming requirements that outline certain procedures that must be followed to ensure proper operation of the TRANS 01-D system.
- An example of a recommended programming style on page 3-2
- Overview of all possible G code functionality that will be available with the TRANS 01-D.

3.1 Application Programming Requirements

Remote operation of the TRANS 01-D via the Operator Interface requires that certain rules for machining, and reverse movements must be established and scrupulously observed by the programmer. This is necessary to insure that NC program execution will always be started in the proper manner, independent of unexpected events and actions, and that the system will always remain controllable in all operating situations via the Operator and Cycle Interfaces. If the programming rules are not followed, the TRANS 01-D will, depending on the type of violation, refuse to issue a “Ready” signal for the start of automatic operation, or it will not be possible to execute a homing command or manual operation selected from the Operator Station. The various programming situations and the requirements for each are summarized in the following sections.

Start of the Program

All machining programs must start with NC Block 000. If several different machining programs are to be written, branching must be accomplished such that an unconditional or conditional jump from NC Block 000 be executed to jump to the start of the NC program.

First Positioning

In order to assure that machining programs will be executed with a correct absolute reference under all circumstances, the first positioning in a machining program should be programmed in Absolute Positioning Mode.

End of the Program

The TRANS 01-D user has the ability to program Incremental and Absolute positional moves. These moves can be executed in NC program blocks With or Without Lag Finishing. The G62 (Without Lag) command in an NC program block is used for velocity contouring between the current NC program block and the next NC program block. Because of this contouring, the last motion NC Block in a profile, must be programmed With Lag Finishing (G61). Additionally, when a NC Block is programmed using Without Lag Finishing (G62), it cannot contain any NC Block jumps.

All NC programs must be terminated with a Jump To Block 000 And Stop command (JS000). This applies equally to machining programs, reverse programs, and tool change programs. It is very important that all NC programs end with a Jump To Block 000 and Stop command. The TRANS 01-D uses this signal for many items, such as monitoring of thermal overloads, Home Switch Monitoring, etc.

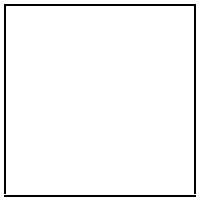
3.2 Programming Capability Description

Up to 200 programmable NC blocks are available from the TRANS 01-D.

3.3 Recommended Programming Styles with an Example

There are two primary modes in which the TRANS 01-D can successfully run its stored NC part program and still be controlled by an external PLC type device, these modes are ***Manual*** and ***Automatic***.

- ***Automatic*** (or ***Auto***) mode is achieved by bringing the TRANS 01-D "Auto" input high (e.g., DEA 4 pin 2 high). The most left column of the CTA10-1 should have "AUTO" running vertically from top to bottom of the screen.

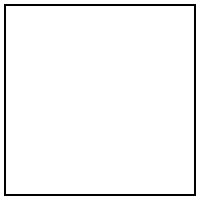


- ***Manual*** mode is achieved by the combination of;

The TRANS 01-D NOT having its "Auto" input high (e.g. IBS object #5FB1 Bit 3 low)

and

The TRANS 01-D has been set to "Single Cycle" mode, number 4 ***Single Cycle*** within the Menu key, and the most left column of the CTA10-1 should have "1CYC" running vertically from top to bottom of the screen.



Automatic Mode

The sample NC part program (Fig. 3-1) on page 3-6 will be referred to when explaining the operation of the TRANS 01-D in "Automatic" mode. The interface selected is a DEA 4 & 5 configuration. This NC part program was written for a three axes servo system with each axis motor containing an *Absolute Encoder* that can be referenced to a positive stop. The NC program demonstrates use of reverse vectors to recover from behind the workpiece. It also shows use of conditional jumps for selection of one of three different NC programs, a move to tool change position, and homing to positive stop.

To run NC part programming blocks N195 to N198, the PLC must have "Auto", "Enable" and "Rev" high and then bring "Home Request" high at least 200msec afterwards.

- The TRANS 01-D is in "Auto" mode at N000 with the "Ready", "At Home" and "No Fault" outputs high.

- The PLC sets the correct Conditional Jumps at the input to the DEA 4 & 5 cards. In this example we select Conditional Jump 3.
- The PLC provides the “Cycle Start” signal at least 50msec after setting the Conditional Jumps.
- The NC program jumps to N010, sets the Reverse Vector to N120, and begins to execute the motion programmed in N011 & N012.

  If at any stage during the execution of N011 & N012 a “Home Request” input is received by the TRANS 01-D it will immediately jump to N120 and begin execution of the absolute positional move to zero of the Y & Z axes at a feedrate of 100 units.
- Once the TRANS 01-D has completed N012, the Reverse Vector is set to N130. The TRANS 01-D continues to execute N013.

  If at any stage during the execution of N013 a “Home Request” input is received by the TRANS 01-D, it will immediately jump to N130 and begin execution of the absolute positional move to zero of the X and then Y & Z axes at a feedrate of 200 units.
- Once the TRANS 01-D has completed N013, the Reverse Vector is set to N140. The TRANS 01-D continues to execute N014.

  If at any stage during the execution of N014, a “Home Request” input is received by the TRANS 01-D it will immediately jump to N140 and begin execution of the absolute positional move forward to the advanced position of the Y & Z axes at a feedrate of 300 units. After reaching the advanced position, it will jump to N130 and begin execution of the absolute positional move to zero of the X then Y & Z axes at a feedrate of 200 units.
- When N014 has been successfully completed, the Reverse Vector is set to the block number (N015) immediately after the block (N014) that contains the JR000 marking to end of the forward profile. The TRANS 01-D continues executing the Reverse Vector program from N015 through N017.

  Any “Home Request” input during these blocks (N015 to N017) would effectively be ignored as the TRANS 01-D considers itself to be executing the current Reverse Vector. It does not back out to the home position – instead it moves forward on the normal return profile until it reaches the home position.
- Once the moves to the returned position are completed, JN180 causes a jump to an exit routine that sets the Reverse Vector to N195 and then does “JS000” to jump and stop at N000. This sets the NC program pointer to N000 and leaves the Reverse Vector at N195 as set in block N180.

Note: JS000 does not set the Reverse Vector to N195. Without the “JR195”, the Reverse Vector would still be set to the block immediately following the block that contained the JR000. Then, after completing block N017, if the “Home Request” were given to the TRANS 01-D, the TRANS 01-D would begin to execute block N015. In cases where different parts of the NC program are selected and executed in series without going back to the returned position, it might be desirable to retain a reverse vector for a special path to recover from the current position and not set it back to the normal homing routine. It is always up to the NC program to set the Reverse Vector.

Manual Mode

The NC part program listed below (Fig 1) will be referred to when explaining the operation of the TRANS 01-D in “Manual” mode. The interface selected is a DEA 4 & 5 configuration. This NC part program was written for a three axes servo system with each axis motor containing an *Absolute Encoder* that can be referenced to a positive stop.

The only way the NC program can run NC part programming blocks N195 to N198 is for the PLC to have input “Auto” low with “Enable” and “Home Request” high and at least 200msec afterwards to bring “Reverse” high.

“Manual” mode is controlled primarily by the “Forward” and “Reverse” inputs of the TRANS 01-D. The “Reverse program” in “Manual” mode refers to the NC part programming blocks that appear after the NC block that contains the “JR000”. Where the “JR000” is programmed is at the discretion of the NC part programmer but it is generally programmed when the machining of the part is complete and the subsequent NC part programming blocks contain motion to return all axis to zero reference.

- The TRANS 01-D is in “1CYC” mode at N000 with the “Ready”, “At Home” and “No Fault” outputs high.
- The PLC sets the correct Conditional Jumps at the inputs to the DEA 4 & 5 cards. In this example we select Conditional Jump 1.
- The PLC provides the “Forward” signal at least 20msec after setting the Conditional Jumps.
- The NC program jumps to N020 - sets the Reverse Vector to N120 – and begins to execute the motion programmed in N021 & N022.

  If at any stage during the execution of N021 & N022, the “Forward” input is brought low and the “Reverse” input is brought high to the TRANS 01-D, it will immediately jump to N120 and begin execution of the absolute positional move to zero of the Y & Z axis at a feedrate of 100 units.
- Once the TRANS 01-D has completed N022, the Reverse Vector is set to N130. The TRANS 01-D continues to execute N023.

  If at any stage during the execution of N023, the “Forward” input is brought low and the “Reverse” input is brought high to the TRANS 01-D, it will immediately jump to N130 and begin execution of the absolute positional move to zero of the X then Y axis at a feedrate of 200 units.
- Once the TRANS 01-D has completed N023 the Reverse Vector is now set to N142. The TRANS 01-D continues to execute N024.

  If at any stage during the execution of N024, the “Forward” input is brought low and the “Reverse” input is brought high to the TRANS 01-D, it will immediately jump to N142 and begin execution of the absolute positional move forward to the advanced position of the Y axis at a feedrate of 100 units. After reaching the advanced position, it will jump to N130 and begin execution of the absolute positional move to zero of the X then Y & Z axes at a feedrate of 200 units.
- When N024 has been successfully completed, the Reverse Vector is set to the block number (N025) immediately after the block (N024) that contains the JR000 marking to end of the forward profile. Holding of the “Forward” input high will no longer allow the continued execution of the block N025. The “Forward” input must be brought low and the “Reverse” input high and the Reverse program from N025 through N027 will now be run.

- Once the moves to the returned position are completed, JN180 causes a jump to an exit routine that sets the Reverse Vector to N195 and then does “JS000” to jump and stop at N000. This sets the NC program pointer to N000 and leaves the Reverse Vector at N195 as set in block N180.

Note: JS000 does not set the Reverse Vector to N195. Without the “JR195”, the Reverse Vector would still be set to the block immediately following the block that contained the JR000. Then, after completing the “JS000” block, if the “Reverse” were given to the TRANS 01-D, the TRANS 01-D would begin to execute block N025. In cases where different parts of the NC program are selected and executed in series without going back to the returned position, it might be desirable to retain a reverse vector for a special path to recover from the current position and not set it back to the normal homing routine. It is always up to the NC program to set the Reverse Vector.

```
N000 JC010:00100                                          // Conditional jump to Two heads cutting
N001 JC020:00001                                          // Conditional jump to Y head cutting
N002 JC030:00010                                          // Conditional jump to Z head cutting
N003 JN180
                                                          // Jump to Exit
…..Two heads cutting ….
N010 JR120                                                // Sets reverse vector to N120
N011 G00 G90 G62 Y3.000 Z3.000 M22222222211               // Rapid heads to part
N012 G01 G90 G61 Y11.000 Z11.000 F112.0 M11222222222 JR130 // Feed heads to cut left walls
N013 G01 G90 G61 X29.405 F78.0 M22220022222 JR140         // Feed main slide to cut back wall
N014 G01 G90 G61 Y5.750 Z5.750 F112.0 M22110022222 JR000  // Cut right walls, end forward profile
N015 M11000000000                                         // Hydraulic acknowledgments
N016 G00 G90 G61 Y0.000 Z0.000                            // Rapid Return heads to home position
N017 G00 G90 G61 X0.000 M00000000000 JN180                // Rapid Return main slide to home position, Jump to Exit

…..Y axis head cutting…..
N020 JR120                                                // Sets reverse vector to N120
N021 G00 G90 G62 Y3                                       // Rapid head to part
N022 G01 G90 G61 Y11.000 F78.0 M12222222222 JR130         // Feed head & cut left wall
N023 G01 G90 G61 X29.405 F78.0 M22220222222 JR142         // Move main slide to cut back wall
N024 G01 G90 G61 Y5.750 F78.0 M22120222222 JR000          // Cut right wall, end forward profile
N025 M20000000000                                         // Hydraulic acknowledgments
N026 G00 G90 G61 Y0.000                                   // Rapid Return Y axis head to home position
N027 G00 G90 G61 X0.000 M00000000000 JN180                // Rapid Return main slide to home position, Jump to Exit

…… Z axis head cutting …..
N030 JR120                                                // Sets reverse vector to N120
N031 G00 G90 G62 Z3.00 M22222122222                       // Rapid head to part
N032 G01 G90 G61 Z11.000 F78.0 M21222222222 JR130         // Feed head and cut left wall
N033 G01 G90 G61 X29.405 F78.0 M22222022222 JR144         // Move main slide to cut back wall
N034 G01 G90 G61 Z5.750 F78.0 M20212022222 JR000          // Cut right wall, end forward profile
N035 M02000000000                                         // Hydraulic acknowledgments
N036 G00 G90 G61 Z0.000                                   // Return Z axis head to home position
N037 G00 G90 G61 X0.000 M00000000000 JN180                // Return main slide to home position, Jump to Exit

//   ****************** Reverse Program used before left wall cut finished**********************
N120 G01 G90 G61 Y0.000 Z0.000 F100.0 M00000000000        // Return heads to home position
N121 G00 G90 G61 X0.000 JN180                             // Return main slide to home position, Jump to Exit

//   ****************** Reverse Program used before back wall cut finished**********************
N130 G01 G90 G61 X0.000 F200.0 M00000000000               // Return main slide to home position
N131 G01 G90 G61 Y0.000 Z0.000 F200.0 JN180               // Return heads to home position, Jump to Exit

//   ****************** Reverse Program used before right wall cut finished**********************
N140 G01 G90 G61 Y11.000 Z11.000 F300.0 M00000000000 JN130 // Move heads to forward position, Jump to back wall recovery

//   ****************** Reverse Program used before Y-axis has back wall cut finished*************
N142 G01 G90 G61 Y11.000 F100.0 M00000000000 JN130        // Move Y head to forward position, Jump to back wall recovery

//   ****************** Reverse Program used before Z-axis has back wall cut finished*************
N144 G01 G90 G61 Z11.000 F100.0 M00000000000 JN130        // Move Z head to forward position, Jump to back wall recovery

//   ************************** Jump on Event Routine ************************************
N170 M00000000011 JN180                                   // Jump on Event routine

//   ************************** Exit Routine (all paths end here) *****************************
N180 JR195                                                // Set Reverse Vector to block N195
N181 JS000                                                // Jump and Stop to block N000

//   ************************** Tool Change Routine ***************************************
N185 G01 G90 G61 Y-7.0 Z-7.0 F100.0                       // Absolute X & Z axes move to Tool Change position
N186 G01 G90 G61 X0 F100 M22222212222 JN180               // Absolute X axis move to Tool Change position, set tool change auxiliary, Jump to
                                                          Exit

//   ************************ Home to Positive Stop Routine *********************************
N190 G69 G90 Y0.000 F30.0                                 // Home to Positive Stop for Y axis
N191 G69 G90 Z0.000 F30.0                                 // Home to Positive Stop for Z axis
N192 G69 G90 X0.000 F30.0 JN180                           // Home to Positive Stop for X axis, Jump to Exit

//   ************************ Homing Routine ***********************************************
N195 M00000000000
N196 JC190:11000                                          // Conditional jump to Home to Positive Stop
N197 JC185:01000                                          // Conditional jump to Tool Change
N198 G01 G90 G61 Y0.000 Z0.000 F135                       // Absolute Y & Z axes move to zero
N199 G01 G90 G61 X0.000 F135 JN180                        // Absolute Y & Z axes move to zero, Jump to Exit
```

Fig. 3-1: Program Example

3.4 TRANS 01-D NC Code Descriptions

In this section, the NC codes, as used by the TRANS 01-D, are described in detail along with illustration to help explain the function. The following table contains the functions that are available in each NC Block, depending on the selected Process, Axis, and Spindle parameters.

NC CODE	FUNCTION	Refer to Page
G00	Rapid Feed Positioning	3-13
G01	Programmed Feedrate Positioning	3-13
G04	Dwell time	3-29
G08	Adaptive Depth	3-16
G20	Re-enable Axis	3-8
G21	Disable Axis	3-8
G36	Rotary Positioning - Shortest Path	3-22
G37	Rotary Positioning - Positive Direction Only	3-23
G38	Rotary Positioning - Negative Direction Only	3-23
G61	With Lag Finishing	3-14
G62	Without Lag Finishing	3-14
G69	Home to a Positive Stop	3-10
G74	Homing	3-10
G75	Enable Feed to a Positive stop	3-15
G76	Disable Feed to a Positive Stop	3-15
G90	Position command - Absolute (Destination)	3-13
G91	Position command - Incremental (Distance)	3-13
X, Y, Z	Axis designation	
F	Feedrate or Dwell Time	3-29
T	Tool Correction register number to be used	3-29
S	SERCOS spindle RPM speed (Must be enabled in Parameters)	3-33
P	Spindle position	3-33
M	Auxiliary functions	3-35
Jx	Program jumps:	3-35
	JN -- Unconditional jump	3-36
	JU -- Jump to subroutine	3-39
	JR -- Reverse vector programming	3-37
	JS -- Jump and stop	3-37
	JC -- Conditional jump	3-36
	JW -- Conditional Jump with a Wait Time	3-39
	JReturn -- Return from subroutine	

Table 3-1: Available Program Functions

Axis Enable and Disable (G20, G21)

This functionality allows the Indramat drive to continue monitoring the actual position of the motor encoder even though there is holding torque applied from the drive. This is commonly utilized with rotary axis applications where hydraulic clamping mechanisms are used to maintain the stability of high mass drive-trains rather than risk motor overload during cutting.

Because there is no torque applied to the motor during G21 (readout from drive will be “Ab”) only dwell times, block jumps and Input / Output handling should be used while the drive is in this state.

The Axis Disable feature needs to be selected in Aa16 of whichever drive is to be programmed with the G21 / G20 command, otherwise the error “ 785: Axis NOT configured for AF switching ” will be generated.

NC Program Example:

N001 G21 Z0

N002 M10000000000

N003 G20 Z0

N004 G91 G61 Z10 F10 M00011100000 .....

Explanation:

N001 G21 (Axis disable) when executed will drop the selected Indramat servo drive, Z axis in the above example, into its **‘AB’** state. This means that this drive is now disabled, it still has control and bus voltage, but will not exert any torque on the motor shaft. The drive system is still monitoring any change of the motor position through the motor encoder. To specify the axis use either Z0, Y0 or X0.

N002 This block signifies whatever is to occur while the servo-drive is in its disabled state. This particular block is waiting for an Acknowledgment Output to be acknowledged before proceeding to the next block. This acknowledgment could be a signal from a mechanism confirming that a physical shot-pin is in place to stabilize a rotary table. These pins may not be aligned up exactly with their respective slots, but they will not encounter any opposing torque from the drive system as they force themselves into their slots.

N003 This block return the drive to its **‘AH’** enabled state and normal motion can now be commanded of the drive. The drive knows exactly where it is after the N002 block and will not try to make up any following error from the N001 block.

N004 The NC part program continues.

Basic Homing Program

When incremental feedbacks are used, a homing program for travel to the reference position is required at NC Block 195. It must conform to the following requirements:

1. No instruction for travel to a particular position is permitted if there is no prior instruction for homing.
2. The NC program must always contain a Homing instruction (Block 195 is Default NC Block).

3. The NC program must always be terminated with a Jump To Block 000 And Stop instruction. The simplest such program in NC Block 195 could be Homing at a feedrate of 100 UPM (Units per Minute) with a Jump to Block 000 and Stop.
4. Only one axis can be programmed in a G74 block

Note: Absolute feedbacks, when used in TRANS 01-D systems, do not require homing, but any profile used to return the axis to its home position should begin in Block 195.

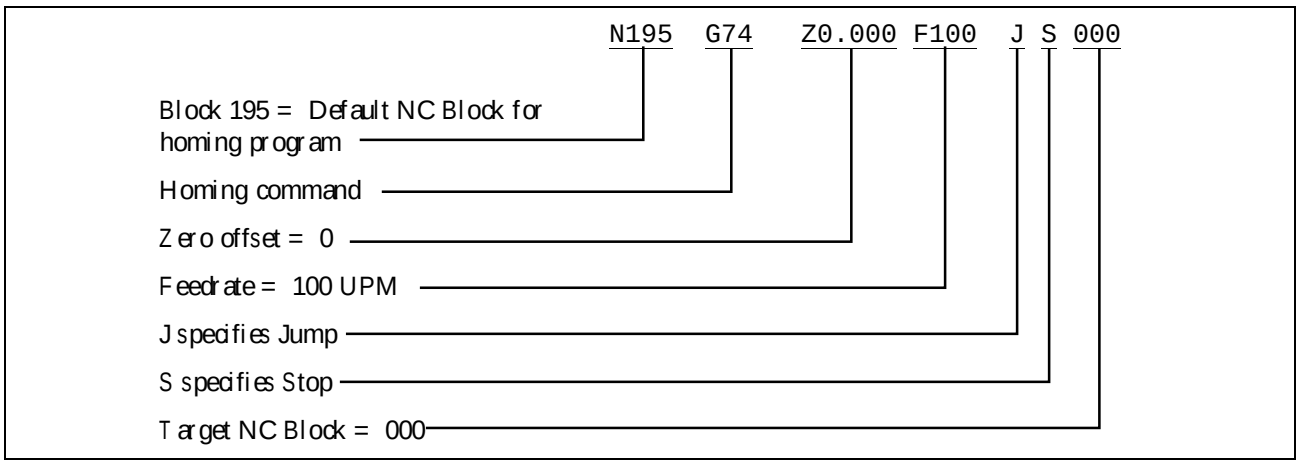

N195 G74 Z0.000 F100 J S 000
Block 195 = Default NC Block for homing program
Homing command
Zero offset = 0
Feedrate = 100 UPM
J specifies Jump
S specifies Stop
Target NC Block = 000

Fig. 3-2: Simple homing instruction

The homing program must be designed so that safe retraction is possible under any condition, including power shutdown. To insure this, the TRANS 01-D has been provided with "reverse vectors" which will determine the NC Block number to which the program will jump if a Homing (Reverse) command is issued in “Automatic” or “Manual” modes.

Note: With TRANS 01-D version 05 but with TRANS 01-D version 06 a JS000 does ***not*** reset the reverse vector to N195 ***but*** leaves the current reverse vector active.

The reverse vector number is retained even if a power failure occurs.

As described in the section titled, "Auxiliary Functions (NC Code M)", auxiliary outputs can be issued at various points in the NC program and the TRANS 01-D waits for an acknowledgment for each output turned on or off before it executes the next NC Block. The one exception to this is that a jump to a reverse program is performed even if the acknowledgments do not match their associated outputs, provided that the first NC Block of that program performs only auxiliary output functions. (This is useful for an emergency return or upon recovery from a power failure.)

This first NC Block in the reverse program (usually 195) should force the auxiliary outputs into a state where they match their acknowledgments. The next NC Block then will usually be a homing command. This should only be performed if it is indeed safe to force auxiliary outputs off and move.

Homing and Zero Offset (NC Code G74 & G69)

G74 Standard Homing

When G74 Homing is selected during dialog programming, the CTA10-1 will display ZERO OFFSET? Note that a reference value is entered as a system parameter (Ax13). This reference value is used to establish the machine reference point as some point other than Home, such as the center point of the slide. If all references to the part are to be

programmed with respect to this machine reference point, a zero offset value of 0 must be entered.

However, if the measurements in the NC program are to be programmed with respect to some other reference, such as the face of the workpiece, the distance from the machine reference point to the workpiece reference point is entered as the zero offset, providing a new reference point which is offset some specified distance from the machine reference point.

The value entered as the zero offset is added to the reference position after homing has occurred. Thus, the zero offset can be used by the programmer in order to program the measurements in a machining program with respect to one of the surfaces of the work piece.

In order to assure correct measuring references, programs that use zero offset referenced measurements must be started with a homing instruction that sets the corresponding zero offset.

Note that the control is at Home when the slide is at the position where the first marker pulse (zero pulse) occurs after closure of the Home Limit switch, and that no movement of the slide will occur when programming a zero offset.

The zero offset provides the flexibility to change the reference point whenever a different part is handled on the TRANS 01-D transfer line, or to correct differences between actual and designed Home position.

Note: For information specific to Homing a rotary axis, refer to the Rotary Motion Control section.

G69 Home to a Positive Stop

Home to Positive stop is a feature used for convenience to initialize the position of an Indramat multi-turn absolute feedback. This is a requirement encountered in the commissioning of machines using absolute feedback devices. When first installed, or whenever the feedback-to-machine orientation is disrupted, the feedback will report a position that is not relevant to the actual machine state. A method must be available to orient the machine to a known position, then load the absolute feedback with that value. SERCOS equipped Indramat drives contain a feature that allows this via the setting of a SERCOS procedure. The SERCOS Ident used is P-0-0012. This function is also available in Indramat's Visual TRANS software, in easy to use graphic screens. G69 offers an alternative method to perform this orientation. It achieves this by moving the slide in the following sequence:

1. The axis moves in a parameter-dictated direction until a stalled-motor condition is detected; that is, a positive stop is found. The length of this move is limited to the total travel distance as defined in the travel limit parameters (Aa06), plus 10%. If the stop is not found in this distance, an error results.
2. The axis reverses direction and moves away from the stop a distance equal to the value programmed in parameter Aa22 ("Home to Stop Distance").
3. When the axis has reached the above position, the absolute encoder value is reset to the value stored in the Reference Position parameter (Aa13).

Note: G69 is intended for use only during initial machine commissioning, or when alterations have been made to the drive train that destroy the relationship between the absolute encoder and the actual machine position. This could occur for example during removal of the motor, gearbox, or ball screw. A move to a positive stop is not always a repeatable function. Changes in drive train temperature, friction, or compliance, as

well as contamination in the area of the stop (such as cutting chips), can cause the relationship between actual slide position and the absolute encoder setting to be different between two home to positive stops. For this reason, axis position should be accurately checked after the operation.

Note: G69 should not be programmed as a machine operator accessible function. It should only be accessible to maintenance personnel. During machine commissioning, it is advisable to disable the function via parameter Aa12 after the absolute feedback is set, to prevent unintentional repeats of the function afterwards. An alternative to this function is to use the absolute encoder initialization routine available in Visual TRANS software.

Note: Do not program a G69 function in the default block for homing (N195). This will cause the TRANS to execute the procedure every time the TRANS-01 is commanded to return, resulting in possible position errors. See warnings above.

Associated Parameters

Aa11: Directions	The homing direction parameter will determine the direction the axis will move when first searching for the positive stop. A '0' programmed in this parameter will cause the axis to search in the "plus" direction. A '1' will cause the axis to search in the "minus" direction.
Aa12: Homing Reference	This parameter indicates the method used to determine the initial home position. Home to a Positive Stop is enabled here, by selecting option '4'.
Aa14: Reference Position	This parameter contains the absolute value that should be used for home position when it is found. The actual position register is loaded with this value at the end of a G69 function.
Aa20: Positive Stop Feedrate	Aa20 contains the maximum feedrate that can be programmed when executing any positive stop functions. This also applies to G69, Home to Positive Stop.
Aa21: Positive Stop Torque %	This two-part parameter indicates the percentage of available torque the motor should be limited to when (1) Approaching the positive stop ("To the Stop"), and (2) once the positive stop is detected ("At the Stop"). These values are used for both Move to Positive Stop (G75) and Home to Positive Stop (G69).
Aa22: Home to Stop Distance	During a G69 function, the TRANS-01D uses this parameter to determine the distance the axis should move away from the positive stop, before stopping and setting reference position. The value is non-signed, as the direction is determined by the inverse of the state of parameter Aa11.

Programming

G69 is programmed similar to any move command.

Required block contents:

Function:	G69 is entered into the block as the block function.
Axis Word (X, Y, or Z):	The block must contain the axis-word for the desired axis. The axis word must have a value associated with it equal to zero (example: "X0"). Zero offsets, such as those available with G74, are not available with G69. Zero Offsets are used to temporarily change axis offsets, as an aid to simplify part programming for multiple parts. G69 is intended for machine commissioning and repair only, and therefore should not be used to temporarily change axis co-ordinates. Only one axis is allowed per G69 operation.
Feedrate (F):	It is suggested that a feedrate always accompany a G69 function in a block for clarity and safety. The feedrate entered must be less than or equal to the value entered in Axis Parameter Aa20 ("Maximum Speed to Positive Stop"), or an error will result during execution. If no feedrate value is entered, the TRANS 01-D will default to the value in Aa20 (max. speed to positive stop).
Lag Finishing:	G69 always operates with G61 ("With Lag Finishing"). An error will result if an attempt is made to execute a G69 function with G62 ("Without Lag Finishing") active. G62 is a modal value, however, it is recommended that G61 always accompany a G69 move for clarity and certainty.

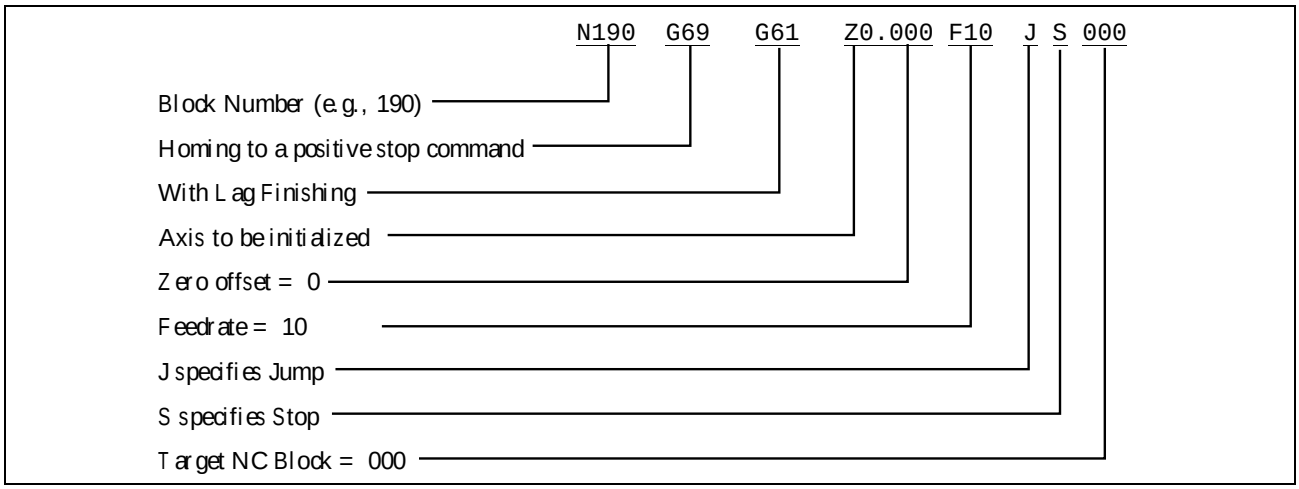

N190 G69 G61 Z0.000 F10 J S 000
Block Number (e.g., 190)
Homing to a positive stop command
With Lag Finishing
Axis to be initialized
Zero offset = 0
Feedrate = 10
J specifies Jump
S specifies Stop
Target NC Block = 000

Fig. 3-3: Example Program Block

Operation Using the previous example (N190 G69 G61 Z0 F10 JS000), with the following parameter conditions:

- Aa06: Overtravel Limits: = +20.000 and -0.400
- Aa11: Directions - Homing = 1 (Find stop in Negative Direction)
- Aa14: Reference Position = 4.000
- Aa22: Home to Stop Distance = 0.500

The motion profile would be as follows:

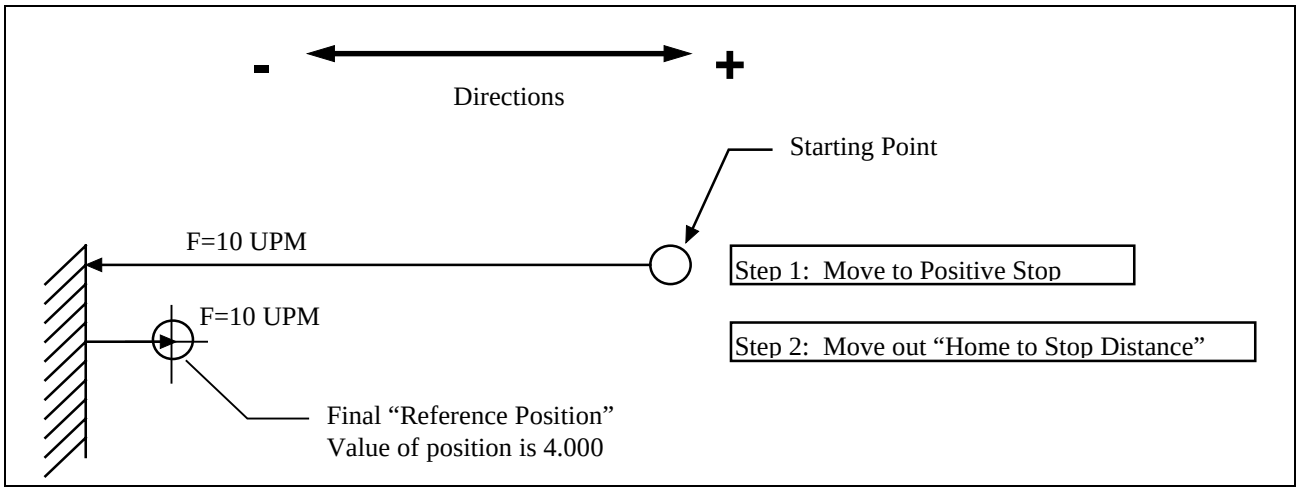


Fig. 3-4: Example Motion Profile

Note: Step 1 will attempt a move of 22.44. ([+20 - -0.4]*1.10). If the positive stop is not found in this range, a "Positive Stop Missing" error will result.

Note: Any subsequent G74 (homing) commands would move to the position defined here as "reference position". Any Zero offset value in them would be added to the value of the reference position parameter after reaching this point, and its value would change to that result.

Positioning (NC Code G00, G01, G90 & G91)

G00 and G01 specify that a positioning move will be executed in this program NC Block. G00 specifies the speed to use for the commanded move will be at the Rapid Speed entered into the axis parameter. In this case, a Feedrate is not required in this NC Block. G01 specifies a positional move, but the programmer is prompted to enter a Feedrate value to be used for this move. If the programmer does not enter a

Feedrate in this NC Block, it will be executed using the last feedrate value used for a positional move.

Two types of positioning can be selected in the system, absolute (G90) and incremental (G91).

Note: For information specific to Positioning a rotary axis, refer to the Rotary Motion Control section.

In absolute positioning (G90), all movements of the slide are made to some absolute distance from the machine reference position, which will either be Home or some offset position from Home. Thus, if the slide is at +2 inches from Home, a command to travel to +3 inches results in a one inch feed in the positive direction.

In incremental positioning (G91), all movements of the slide are made in the commanded direction to the distance specified, starting from the current position of the slide. Thus, if the slide is at +2 inches from Home, a command to travel +3 inches incrementally results in the slide positioned at +5 inches from Home.

NC Program example:

```
N001  G01  G90  G61  Z10  F20   M11122211122
N002  G00  G90  G61  X10  M22211122211
N003  G00  G91  G61  X20  M00022200022
N004  G01  G90  G61  X0   Z0   F20   M11111111111
```

Explanation:

N001 Absolute move with lag to Z10 at a feedrate of 20 and Aux 1,2,3,7,8,9 are turned on

N002 Rapid absolute move with lag to X=10 and Aux. 4,5,6,10,11 are turned on

N003 Rapid absolute move with lag to X=20 and Aux. 1,2,3,7,8,9 are turned off

N004 Absolute interpolated move with lag of X and Z to 0 with a path feedrate of 20 and Aux. 1,2,3,4,5,6,7,8,9,10,11 are turned on.

With / Without Lag During Positioning (G61 & G62)

When a position command is issued, the servo motor moves the axis in response to that command. There will always be some finite lag time between the time the command is issued and the time the servo motor brings the axis into position. It is important to note that, in a program NC Block, the TRANS 01-D does the positioning first, then performs any miscellaneous functions such as jumps or turning auxiliary functions on or off when it finishes the movement.

When programming your positioning commands, you will be required to respond to the "With/Without Lag?" display.

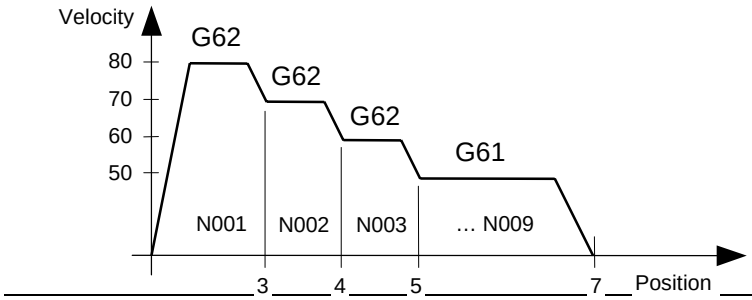
With Lag Finishing (**G61**) specifies that the axis must be in position before any miscellaneous functions remaining in the NC Block are executed or before the next NC Block is executed. This would be required at full depth, for example. It is important to note that this is also required where you have programmed miscellaneous functions, such as auxiliary outputs (see the section titled "Auxiliary Functions (NC Code M)" for more information) which are to turn on only when the axis is in position.

Without Lag Finishing (**G62**) specifies that the velocity profile will be contoured from one NC Block's feedrate to the next NC Block's feedrate

to avoid a stoppage of motion between NC Blocks. The position programmed in a G62 NC Block will be the position at which the axis has reached the next NC Block's feedrate. The move is considered to be finished once the axis is accelerating/decelerating into the subsequent feedrate, but prior to the time the position is actually reached. Thus, any auxiliary functions in this NC Block may be turned on while the axis is still in motion. Therefore, With Lag Finishing may not be necessary to your positioning operation, but you may need to select it to insure that auxiliary functions are not turned on too soon.

Note: When programming G62 "without lag" moves, a total of nine (9) NC program blocks can be used to complete one motion profile. However, the motion profile must end with a G61 "with lag" NC block included as one of the nine NC blocks.

Example:
N001 G01 G90 G62 X3 F80
N002 G01 G90 G62 X4 F70
N003 G01 G90 G62 X5 F60
……
N009 G01 G90 G61 X7 F50



Enable/Disable Feed To A Positive Stop (G75 & G76)

This function may be used when it is necessary to position the slide against a positive mechanical stop. The slide will move at the feedrate programmed in this block. The available torque of the motor will be reduced to the percentage value specified in parameter Aa21, "% Torque To Pos Stop". When the Trans 01-D senses that the motor has stalled, the motor's available torque will be changed to the percentage value programmed in parameter Aa21, "% Torque at Pos Stop". This torque value will still be used for any Dwell or other waiting period, i.e., auxiliary function acknowledgments. The torque value will be switched back to its previous value when the Trans 01-D executes a G76 program command.

The distance (G91 incremental) or destination (G90 absolute) programmed with this function is the maximum distance the slide will be allowed to travel and should be a point just past the expected positive stop. If the slide reaches this position without the motor stalling, movement will stop, the diagnostic POS STOP MISSING will be displayed and a soft fault will result. To recover, you must press the CE (Clear Error) key. An incremental distance (G91 command) programmed in a block following a feed-to-positive-stop will be based on that point where the stall occurred.

If the theoretical programmed position is too close to the positive stop, the error message 'Positive Stop Missing' will be displayed.

The TRANS-01-D will recognize the positive stop in two ways.

1) The feedback velocity falls below 1% of the commanded velocity and

2) the torque value exceeds the pre-set value in parameter Aa21 - % Torque to the Stop for 48 msec.

If both of the aforementioned situations occur together for 120 msec, the TRANS 01-D will consider the positive stop as found. It will then reduce the drive's torque level to the value set in Aa21 - % Torque at the Stop.

When the theoretical end position of the G75 block is programmed, the theoretical end position should be at least four times the following error past the mechanical positive stop. The following error is calculated as follows: Following Error = Programmed speed in G75 block/Kv factor (Aa08) * 1000.

NC Program example:

```
N039  G01  G90  G61  Y18  F200  M11100022211
N040  G75  G90  Y21  F112        // Feed to Pos Stop
N041  M00000022200               // AUX I/O
N042  G76  G90  G61  Y0   JS000  // Disable Pos Stop mode
N195  G76  G90  G61  Y0          // Ensure Pos Stop is disabled
```

Explanation:

N039 Absolute move with lag to Y=18 at a feedrate of F=200 and Aux. 1,2,3,10,11 are turned on and Aux. 4,5,6 are turned off.

N040 Feed to a positive stop at Y=21 at a feedrate of 112.

N041 When a positive stop has been achieved, Aux. 1,2,3,4,5,6,10,11 are turned off.

N042 Disables positive stop mode and returns nominal to the Y axis

N195 Disable positive stop mode (whether enabled or not) on the Y axis before executing the rest of the homing command.

Adaptive Depth Control (G08)

Adaptive Depth arises from the final depth of a positional move being dependent on the location of the part surface and not the referenced position of the motor encoder. This is made possible by using an external encoder to determine the final position. This form of positioning has the following advantages:

- It compensates for both drive train and work piece variations while the tool is actually performing the cut.
- Using incremental positioning it's possible to program distances relative to the face of the part.

The TRANS 01-D begins adjusting the final depth of a positional move using the secondary device when a G08 (Adaptive Depth Command) is issued in the NC block of the part program. This G08 command must be preceded by a G62 (without lag command).

In an ideal situation, once a G08 command has been issued the drive will continue to move until the position reading from the external encoder is equal to that of the G08 command. But the final position also takes into account whatever minimal deflection that may have existed (e.g. due to vibration, mechanical binding etc.) on the ext. encoder. To do this the TRANS 01-D takes a snapshot of the ext. encoder feedback position when it reaches the first NC block containing a G62 move preceding a G08 NC part. In reality the final positional reading of the ext. encoder equals the deflection seen on the ext. encoder when the first G62 NC block preceding a G08 was issues *plus* the distance commanded in the G08 NC block.

The final destination point of this G62 move must be such that the ext. encoder has been depressed by at least 50 micro-meters and not more than the value set in TRANS 01-D axis parameter Aa31(Linear Encoder Pre-Limit) .

If the value of the position in the final G62 block plus the position value located in the G08 block is greater than the value in TRANS 01-D axis parameter Aa06 an over travel fault ' Position out of bounds ' will occur.

Once the TRANS 01-D is in any other NC block other than one that contains a G08 command then the ext. encoder is not being used for positioning.

Hardware and Software Requirements

Indramat Servo Drive	DIAX 03/04 Type Drive
Firmware	TRANS 01-D 06V11 or higher DIAX 03 Drive DSM 2.2 SSE 02V26
Indramat Interface Card	DLF (latest revision)
Indramat Encoder Cable	Type 03-0349
Online Programming Tool	Indramat Visual TRANS software
External Linear Encoder	Heidenhain MT25W (incremental measuring device)

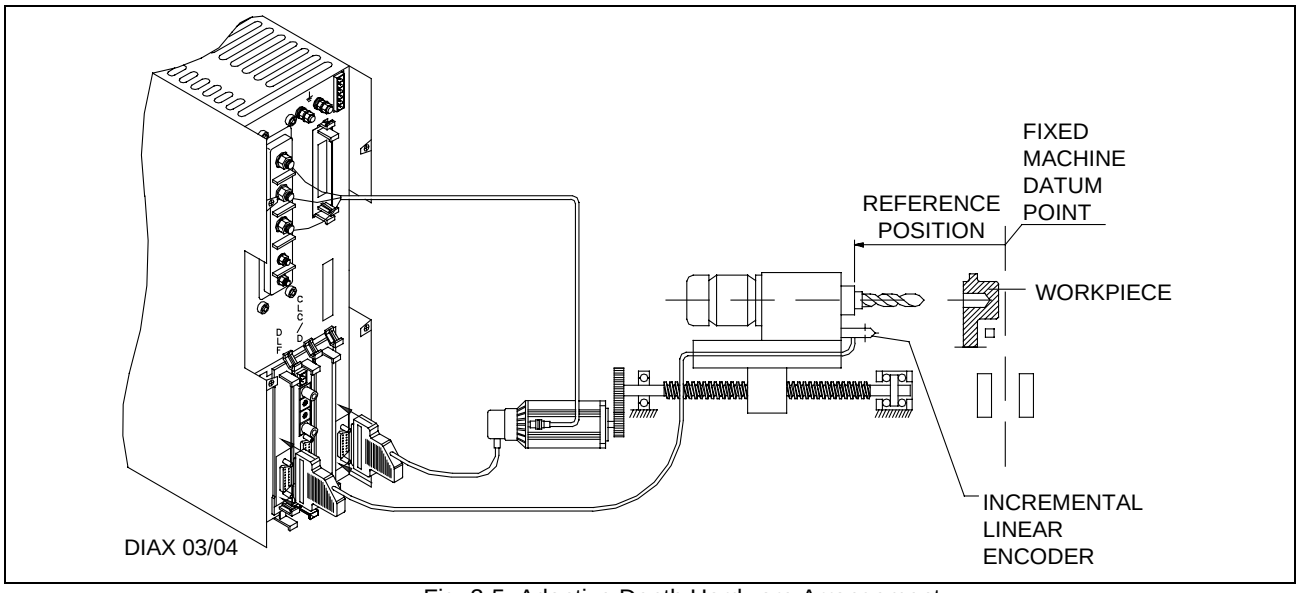


Fig. 3-5: Adaptive Depth Hardware Arrangement

Programming Example

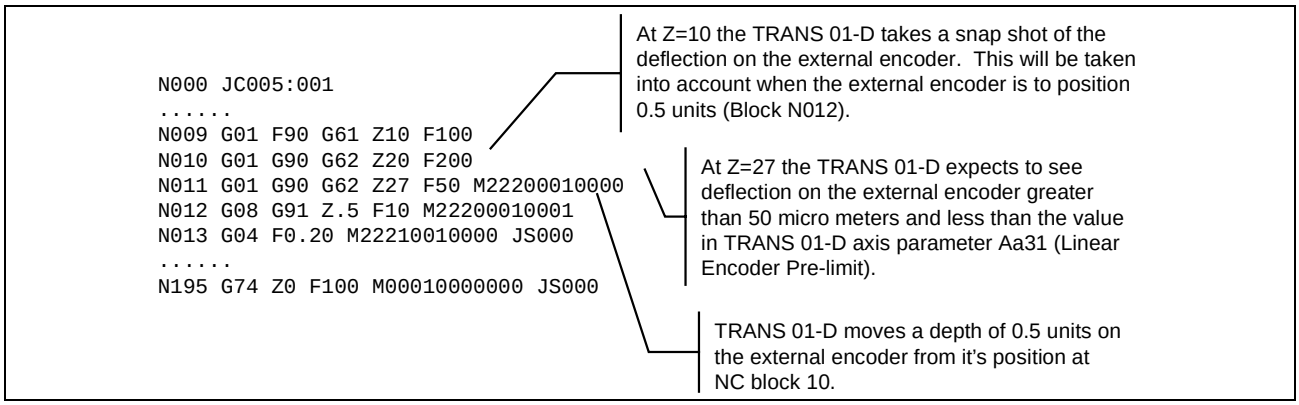

N000 JC005:001
.....
N009 G01 F90 G61 Z10 F100
N010 G01 G90 G62 Z20 F200
N011 G01 G90 G62 Z27 F50 M22200010000
N012 G08 G91 Z.5 F10 M22200010001
N013 G04 F0.20 M22210010000 JS000
.....
N195 G74 Z0 F100 M00010000000 JS000
At Z=10 the TRANS 01-D takes a snap shot of the deflection on the external encoder. This will be taken into account when the external encoder is to position 0.5 units (Block N012).
At Z=27 the TRANS 01-D expects to see deflection on the external encoder greater than 50 micro meters and less than the value in TRANS 01-D axis parameter Aa31 (Linear Encoder Pre-limit).
TRANS 01-D moves a depth of 0.5 units on the external encoder from it's position at NC block 10.

Fig. 3-6: Adaptive Depth Programming Example and Sequence

Whatever the amount the encoder is deflected (e.g., by λ) as it begins to execute NC block N012, the TRANS 01-D will move the axis a distance of (0.5 - λ) under adaptive depth control to equal 0.5 total linear deflection at the end of NC block N012.

Note: If motion on the probe occurs after the G62 command preceding the G08 has been issued, but before the probe has come in contact with the part then this positional variation will not be accounted for by the TRANS 01-D. But this error will be accounted for once the probe moves onto the part (unless there is physical damage to the external encoder).

Set-up Procedure

1. Ensure that the DIAX 03/04 drive system is powered off. Install the DLF board.
2. Attach the Heidenhain External Encoder to the DLF board using the Indramat IKS0349 cable.
3. Power up the DIAX 03/04 drive system.
4. Start communicating with Visual TRANS (using RS232) through the serial port " X27 " on the CLC-D card on the DIAX 03/04 drive.
5. Using Visual TRANS check that SERCOS parameter " S-0-0030 Manufacturer Version " displays the correct version of DSM - "DSM 2.3-SSE-02V26" or higher.
6. CLC-D card parameter # " C 0-0306 In position window " needs to be set to " .001mm " to ensure that the DIAX 03/04 drive and ext. encoder home correctly.

Configuration Procedure

1. Put the TRANS 01-D into parameter mode.
2. TRANS 01-D axis parameter Aa01, number 2 needs to be set to "1" to activate the G08 (Adaptive Depth Command) in the NC part program block. If a G08 is used in an NC program without this parameter activated the following error appears: "779 Adaptive Depth not configured for this axis"
3. TRANS 01-D axis parameter Aa03 (Feed Constant) needs to contain an accurate value, of the axis ball screw that the Adaptive Depth is being programmed on, as this value is used to ensure consistency between the motor encoder feedback and the ext. encoder feedback.
4. TRANS 01-D axis parameter Aa30 (Adaptive Depth Max Speed) needs to be set to the maximum allowable velocity when programming a G08 (Adaptive Depth Command). *When using Visual TRANS, this parameter corresponds to axis parameter # 317.* If a value greater than this parameter is programmed in a G08 command block the following alarm appears: " 780 Maximum Adaptive Depth feedrate exceeded "
5. TRANS 01-D axis parameter Aa31 (Linear Encoder Pre-limit) contains the value that the TRANS 01-D will compare to the ext. encoder position at the beginning of the G08 NC block. If the position from the ext. encoder is greater than axis parameter Aa31, the TRANS 01-D will move to the position within the preceding G62 block *plus* the Pre- limit deflection value and then the following alarm appears: " System: 511 Adaptive Depth Pre-Limit Error "
6. Using Visual TRANS make SERCOS parameter " P 0-0075 Interface Feedback 2 " equal to " 2 ". This configures the DLF board as the source of secondary feedback.
7. Exit from Parameter Mode.

8. To ensure that the External Encoder is feeding information back to the DIAX 03/04 drive, push the tip of the MT25W in & out. The position display on SERCOS parameter " S-0-0053 Position Feedback Value 2 Ext. Feedback " should change correspondingly.

Alarm / Diagnostic Number	Cause	Action Required to Reset Condition
425	"Task B: 425 Depth: Probe reading > w3; not zeroed (30)" When referencing the motor encoder (performing a G74) the ext. encoder was not set to a value less than 50 micrometers.	Ensure that CLC-D card parameter C-0-0 306 In-Position Window is less than 50 micrometers.
511	"Adaptive Depth Pre-Limit Error " The ext. encoder is deflected more than TRANS 01-D axis parameter Aa31 at the beginning of the G08 part.	1. The part may be out of tolerance and this is a correct diagnostic. 2. The Pre-Limit value in TRANS 01-D axis parameter Aa31 is too small. 3. There is some mechanical blockage with the ext. encoder.
512	" Adaptive Depth Part Not Found " The TRANS 01-D has reached the position commanded in the G62 block preceding the G08 command plus the Pre-limit value and still has not detected more than 50 micro meters deflection on the ext. encoder.	1. The part may be out of tolerance and this is a correct diagnostic. 2. Test to ensure data is being read from ext. encoder by monitoring SERCOS parameter S 0-0053 when ext. encoder is in motion. There is an error in reading the ext. encoder data. Problem with ext. encoder or Feedback cable or DLF card
779	" Adaptive Depth not configured for this axis " A G08 was issued in an NC part program block without TRANS 01-D axis parameter Aa01 being enabled for Adaptive Depth programming.	Make TRANS 01-D Axis parameter 1 such Adaptive Depth programming is enabled.
780	"Maximum Adaptive Depth feedrate exceeded " The feedrate programmed in the G08 is greater than the value in TRANS 01-D axis parameter Aa30.	Decrease the feedrate less then the value in TRANS 01-D Axes parameter # 30 or increase this parameter greater than the value on the G08 command - depends on the mechanical limitation of the system.
781	"Maximum Adaptive Depth deflection exceeded" The distance traveled by the ext. encoder is greater than TRANS 01-D axis parameter Aa32	Either decrease the distance in the G08 NC block or increase the value in TRANS 01-D axis parameter Aa32 - need to consider mechanical limitations.
813	"G08 depth is less than or equal to Adaptive depth pre-limit"	"The value of the axis depth (distance of travel) in the G08 block is less than or equal to the Pre-limit value located in axis parameter Aa31. Example block: G08 G91 G61 Z**10** M10000000000 Axis parameter Aa31 value: **11**

External Feedback Devices - Distance Coded Linear Scale

Note: **This information applies only to TRANS 01-D software versions 06V11 and later.**

For the use of Distance Coded Linear scales, the operation of the TRANS 01-D will remain the same except for the following cases:

- When referencing for the first time after powering up or in those instances where the axis has lost its reference.
- When executing a Home command (G74) when the axis is referenced
- When the user has made changes to the Reference position since the last time it was Homed or Referenced.

Relevant parameters for Distance Coded Linear scales include:

- S-0-0115 Position Feedback Type (set second LSB to 1 to set distance coded linear scale as feedback type for servo drive)
- S-0-0118 Resolution of Linear Feedback
- S-0-0165 distance coded linear scale Distance-coded Reference Dimension 1
- S-0-0166 distance coded linear scale Distance-coded Reference Dimension 2
- S-0-0178 distance coded linear scale Absolute Offset 2

Operation of Servo Drive with Distance Coded Linear Scale

The Distance Coded Linear scale is a hybrid device that initially acts like an incremental feedback device, but once it is referenced to the machine position grid, it will act as an absolute feedback device. The advantage of the Distance coded scale is the user does not have to re-establish a Home position, it only needs to establish where it is in relation to the machine reference point if axis reference is ever lost.

On power-up or when axis has lost its reference, the system will not be referenced. S-0-0403, LSB will be 0. When a Drive Generated Homing Command (G74) is issued, the axis will traverse the distance necessary to pass two marker pulses, in the direction specified in the Homing Parameter Aa11 as the Homing direction. After it has traversed this distance, it will stop moving and display its absolute position as defined by the scale. At this time the servo drive will set S-0-0403, LSB to a 1 to signal that the axis is referenced. Any subsequent Drive Generated Homing Commands (G74) that are issued will be ignored by the drive.

TRANS 01-D Operation

The TRANS 01-D treats the Distance Coded Linear scale as an incremental feedback device in those cases where the axis will loose its reference, i.e.: Exiting parameter mode, certain drive errors, or when the system has lost all power. In these cases, if reference is lost, the G74 command initiates a Drive Controlled Homing Procedure to once again reference the scale to the machine slide.

If the axis has not lost its reference, the G74 command moves the slide to the already established Home position. When the axis has been moved to its established Home position, the Home output from the TRANS 01-D Cycle interface goes high.

If the user has changed the value of the TRANS 01-D Reference position by changing the value of the axis word in the Homing Block, the system does not re-establish the Home position, unless it has lost reference. However, it does move the axis to the newly designated Home position and output the Home signal to notify the line control that the axis is at the

Home position. If the Reference position parameter value has been changed, the TRANS 01-D once again issues a Drive Generated Homing Command to re-establish its Home position.

The user should enter the value to be displayed when at the Home position using the already established methods, Axis word in the G74 Homing block and by entering a value into the Reference Position parameter (CTA10-1 Aa13 or Axis parameter A-0-0318). These two values are summed and placed into S-0-0054 (Position Feedback 2, Reference Distance) as the value to be displayed when at the Home position.

Rotary Motion Control

The TRANS 01-D has the ability to control a rotary application. This option can be used for applications such as rotary tables or lift and transfer drives. The software will allow programming rotary positions in UNITS/TABLE REVOLUTION. Rotary speeds are also entered in these same units.

Note: The TRANS 01-D can only be configured to control one rotary axis. When used in a multi-axis system, only one of the axes may be rotary.

Rotary software uses conventional TRANS 01-D programming techniques. Only certain motions and/or working units change. The NC program commands G20 and G21 are functional in rotary operation.

Associated Parameters

Since the system of units for rotary motions is arbitrary, the UNITS parameter gives the user the option of specifying the UNITS/TABLE REV to be used for his application. The software and/or hardware Travel Limits can be used to limit the motion to less than one revolution. As an example, the motion on a typical rotary crank arm transfer drive should not travel greater that 180° for its total travel. Using the Travel limits, any motion outside of 270° - 90° will cause an Overtravel fault.

The standard parameters used for Rotary operation are described below.

Parameter	Description
A02	UNITS /TABLE REVOLUTION (set to 360 degrees)
A05	GEAR RATIO
A10	SPEEDS -- Homing Speed, Rapid Speed, Jogging Speed, Jogging Rapid, Max Cutting and Max Speed to Positive Stop are all expressed in unit/table rev/min.
A11	DIRECTIONS: Homing Direction: Determines the direction the axis will move when commanded to Home. Program Direction: This parameter will determine what direction the motor will turn when using G37 and G38 G-codes.

Homing and Zero Offset (NC code G74)

A “0” or a “1” in the Homing direction parameter will cause it to Home in only one direction. When Homing is selected during programming, the TRANS 01-D will display “ZERO OFFSET? “. The value entered as the zero offset will determine the value of the home position on the absolute grid of the table. For example, if degrees are used as the units (360 units/table rev), and 10 is programmed as the zero offset, home position will be called "10 degrees". If an absolute move to zero degrees is commanded after that, the table would move 10 degrees. No additional motion will occur with the inclusion of a zero offset. It is simply a value to be used for home position once it is reached. If a value other than zero is

programmed in the parameter for reference position, this value will be added to the zero offset in the homing block to determine the value for home. Using the above example again, if -25 was programmed as the reference position, home would be called -25 +10, or -15 degrees. Since the display will only show positive values, this would appear as 360-15, or 345 degrees.

Positioning (NC code G90, G91)

When an axis is designated as rotary in the servo drive setup, all positional moves will be made according to the modulo value specified. A rotary axis can be interpolated along with a linear axis. Positioning of the rotary axis can be performed as either absolute positioning (G90) or Incremental positioning (G91). The G90 and G91 commands are modal in the rotary mode of the TRANS 01-D.

G90 This is the default mode for rotary operation. By default, all G01 rotary motion commanded in the TRANS 01-D will be absolute, using the shortest path for the move. When a positional move is commanded, the axis will move to the programmed destination using the shortest possible path, positive or negative direction, whichever is less than one-half of a table revolution away from the commanded position. The destination may be any value between 0 and the number set in the parameters as UNITS/TABLE REV. No negative values are allowed. To specify a direction for the G90 absolute move, in the cases where the shortest path is not desirable or possible, the user has the ability to use two additional G-codes. These two G-codes, G37 and G38 can be used to specify what direction the axis must take when executing the programmed move.

G91 This command for a rotary axis does not require an additional rotary operation G-code. The G01 positional move command will cause the rotary axis to move the incremental distance specified in this NC program block. This type of move will cause the axis to move the specified distance from its present position. Direction may be specified by programming the distance as a positive or negative value. The value of the distance is limited to plus or minus the value for the maximum travel distance of the digital drive system in modulo mode.

Note: The G91 command is not valid for G36, G37 nor G38 moves.

NC code G36

This modal command can be used to specify that the ensuing absolute moves for the rotary axis should use the shortest path-positioning mode. This is the default mode for all absolute moves. If all absolute positional moves to be made will be made using the shortest path, this command will not be necessary. This command is necessary when the user has completed a move using either the G37 or G38 command and they want the next series of moves to be executed using the shortest path mode. In that case, the user must program the first absolute move that is to use the shortest path using this command. This modal command will then be used for each succeeding NC program block until it is changed by programming a G37 or G38.

NC code G37

This G-code gives the user the ability to program an absolute (G90) move inside of the modulo value that is greater than 180 degrees away from their current position and choose positive as the direction the axis should travel to that position instead of the axis taking the “shortest path”.

G90 When a positional move is commanded, the axis moves to the programmed destination in the positive direction only. The destination may be any value between 0 and the number set in the parameters as UNITS/TABLE REV. If travel limits are enabled, a programmed move to

a destination outside of the specified travel range will generate a travel limit violation error.

Note: G91 type positioning is not supported for use with the G37 command.

NC code G38

The intent of this G-code is to give the user the ability to program an absolute (G90) move inside of the modulo value that is greater than 180 degrees away from their current position and choose negative as the direction the axis should travel to that position instead of the axis taking the "shortest path".

G90 When a positional move is commanded, the axis will move to the programmed destination in the negative direction only. The destination may be any value between 0 and the number set in the parameters as UNITS/TABLE REV. If travel limits are enabled, a programmed move to a destination outside of the specified travel range will generate a travel limit violation error.

Note: G91 type positioning is not supported for use with the G37 command.

Rotary Axis Programming Example

This is a programming example for a Rotary Axis using G36, G37, G38 and setting over travel limits in the CTA10-1 / BTC-06 Axis parameter Aa06.

Before setting the valid range of motion for a rotary axis, the user must first determine his range in reference to a unit (0°-360°) circle.

On a unit circle, 0° is located on top. Moving in a clockwise direction from 0° is positive, while counter clockwise is negative.

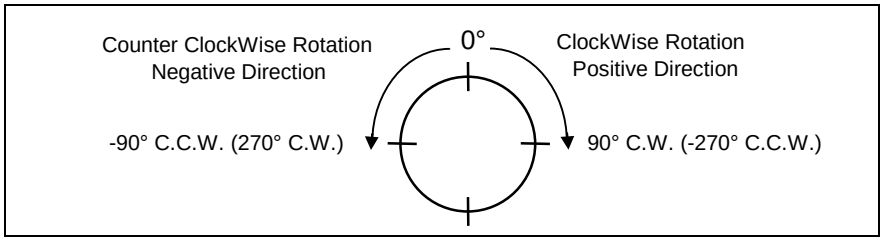


Fig. 3-7: Unit circle rotation

In Fig. 3-1, the range of motion is from **-40º to 71º.**

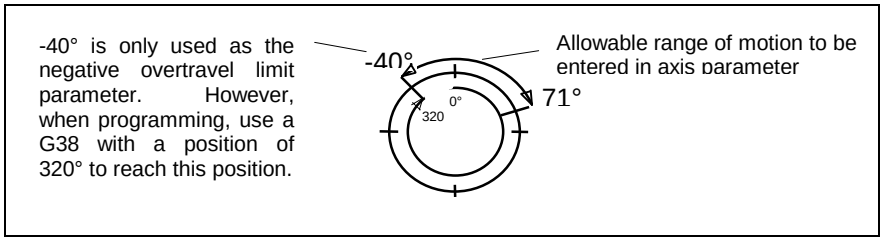

-40° is only used as the negative overtravel limit parameter. However, when programming, use a G38 with a position of 320° to reach this position.
-40°
Allowable range of motion to be entered in axis parameter
71°
0°
320

Fig. 3-8: Valid range of motion for rotary axis

Setting of Overtravel limits.

To set the values in the CTA10-1 axis parameter Aa06:

Determine the max positive travel limit parameter. In this example its **71º**.

Go in a counter clockwise direction to determine the negative travel limit, i.e., enter **-40º** *not* **320º**.

The positive overtravel limit value in axis parameter Aa06 is **71º**, and the negative overtravel limit is **-40º**. Once these values are set, the allowable range of motion for this rotary axis must be between **-40º to 71º.**

The following figure contains additional examples of "range of motion" for entry of axis parameter Aa06.

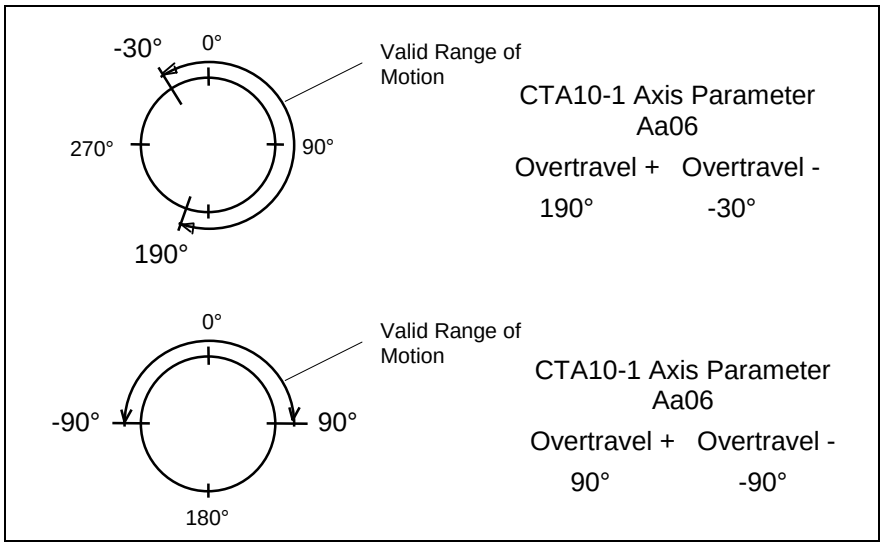


Fig. 3-9: Additional examples of Overtravel range of motion settings

```
N000 M00000000000 JR195
N001 JC010:00001
…….
N008 JS000

N010 M00000000001 JR031
N011 G01 G90 G61 G37 X70.0 F800          // Move Rotary Axis clockwise direction to position 40º
N012 G01 G90 G61 Z20.0 F500 JR024        // Move Linear Axis to position 20 and set reverse vector to N024
N013 G01 G90 G61 G38 X321 F700 JR033     // Move Rotary Axis counter-clockwise to position 320º and sets the Reverse Vector to N033
N014 G01 G90 G61 Z60.0 F500 JR037        //  Move Linear axis to position 60 and set reverse vector to N037
N015 G01 G90 G61 G36 X0.0 F900 JR195     // Move Rotary axis, using the shortest path, to position 0 º and set Reverse Vector to N195
N016 G01 G90 G61 Z0.0 F500 S0 JS000      // Move Linear Axis to 0, Jump and Stop to N000

N195 G74 X0 F720
N196 G74 Z0 F100 JR195
N197 JS000
```

Fig. 3-10: Rotary programming example

Clamping using G20/G21 program commands

When the cutting force exerted on a rotary table is high, it is possible for the position of the drive train to shift. For this reason, many machine builders will clamp the drive train to prevent unwanted movement. If it is not, the system may attempt to hold a position, or even move the drive

train, causing servo overload. The NC program command G21 is functional in rotary mode to accommodate this need. This NC program command should be used for no other purpose than clamping or braking the drive train. If G21 is commanded during NC program execution, the AC Servo Controller Enable will be removed, the drive will go from AF to Ab, allowing free movement of the servo motor (no servo lock). A programmed Auxiliary output can also be used to notify the line control when it is safe to energize the clamp or brake. The Controller Enable will be restored when the G20 NC program command is executed. During the time that the G21 command is in operation, no movements should be commanded, otherwise an error will appear on the display, resulting in a fault. Only dwell times, block jumps, or waits for acknowledgments should be performed while the G21 command is active.

While the G21 function is in operation, the destination will be made equal to the actual position, so that no jump will occur if the motor is rotated. This means that the next positioning command made after a clamping operation should be an absolute move. Incremental moves would be made referenced to the present position (including any move made while clamped) rather than the last commanded position. If incremental moves are required after a move to positive stop, they should be preceded by an absolute move to the last position, to correct for any shift in position during clamping.

The last move made before a clamping operation should be programmed with lag finishing (G61), so that the motor is stopped in position before the Controller Enable is removed. If the block is programmed without lag finishing, the commanded position may not be reached before power to the motor is dropped.

Typical sequence of events for Rotary Operation:

1. The NC program is running and the table is being positioned.
2. Table is in position (G61 move). Auxiliary output is turned on to signal the Line Control that the table is ready to be clamped.
3. Table is clamped through Line Control logic. Ack. is sent to TRANS 01-D signaling table is clamped.
4. Execute the G21 NC program command. TRANS 01-D brings servo drive from AF to Ab and sets servo motor brake. Turn on programmed Aux. output to signal that the drive is disabled.
5. Machining takes place.
6. Machining is completed. An Ack. is sent to the TRANS 01-D signaling machining is complete.
7. Execute a G20 NC program command. TRANS 01-D brings drive from Ab to AF and releases brake. Aux. Output is turned on to signal table is ready to be unclamped.
8. Table is unclamped through Line Control logic. An Ack. is brought low to confirm table is unclamped.
9. TRANS 01-D begins execution of the next NC program block.
10. Ensure that block N195 re-enables all configured axis. i.e., Program a G2.

Rotary Axis examples - Feedrate Interpretation

Feedrates used in conjunction with a rotary motion are interpreted based upon the presence, or lack of, an accompanying linear motion. Three cases can occur:

1. If a rotary axis appears alone on an NC block (i.e., no linear axes programmed), the feedrate is interpreted as angular velocity (modulo units/minute).

Example:

* Rotary Axis = Z

* Module units for Axis Z = 360 units / rev

N000 G01 G90 G61 Z180 F100

In this example, F100 is interpreted as 100 degrees / min (100 modulo units / minute)

2. If rotary and linear axes appear on the same NC block and the linear change in position is zero, the rotary axis will run at maximum velocity (i.e. A-0-0020 Maximum Velocity)

Example:

* Rotary Axis = Z

* Linear Axis = X

* Module units for Axis Z = 360 units / rev

* Units for Linear Axis = Inches

N000 G01 G90 G61 X10 Z180 F100 (X is already at 10)

-or-

N000 G01 G91 G61 X0 Z180 F100

In both of these examples, since the axis is already at the commanded destination, the F100 will be ignored, and the axis will run maximum velocity (i.e. A-0-0020 Maximum Velocity).

3. If rotary and linear axes appear on the same NC block and the linear change in position is nonzero, the feedrate is interpreted as linear velocity and the rotary axis velocity is slaved to the linear motion. Slaved means the rotary speed will be proportioned so that the rotary axis finishes it's move at the same time the linear path is completed. If the resulting speed of the rotary axis would be greater than maximum velocity (i.e. A-0-0020 Maximum Velocity), the rotary axis' speed will be limited to that value.

Example:

* Rotary Axis = Z

* Linear Axis = X

* Module units for Axis Z = 360nits / rev

* Units for Linear Axis = Inches

* A-0-0020 = 2000 degrees / minute

CASE (1)

N000 G01 G90 G61 X10 Z180 F100 (X is at 0)

Feedrate is interpreted as 100 Inches / min. X will complete it's move in 6 seconds, so Z will be run with 1800 degrees / minute ((180 degrees/6)*60).

CASE (2)

N000 G01 G90 G61 X5 Z180 F100 (X is at 0)

Feedrate is interpreted as 100 Inches / min. X will complete it's move in 3 seconds, so Z is calculated to run with 3600 degrees / minute ((180 degrees/3)*60).

Feedrate (NC Code F)

In order to enable position commands, a feedrate must be programmed. It is good practice to always enter a feedrate in a NC Block, unless program operation dictates a NC Block without a feedrate. Feedrate values are modal. If a feedrate is programmed in a NC Block, it will be used for any positional moves in each subsequent NC Block, unless the feedrate is changed in a subsequent NC Block. Feedrate values up to the rapid speed parameter's value (Ax10) may be programmed.

Dwell (NC Code G04)

A dwell is programmed to allow time for some action to occur, such as a dwell programmed after a forward cutting motion to allow a drill to clean the hole and prevent burrs.

Dwell times can be programmed from 0.01 to 99.99 seconds.

Tool Corrections (NC Code T)

The TRANS 01-D includes a feature that allows programmed corrections to be entered to compensate for changes in the tool or part dimensions. Program entry is in a two-digit tool correction register, which has an associated correction (or compensation) value. Nine (9) Tool Correction registers are available per servo axis, plus additional registers for External Tool Correction. The registers for each axis are;

Axis	Internal Register	External Register
X axis	T11 - T19	T01
Y axis	T21 - T29	T02
Z axis	T31 - T39	T03

The values in registers Tx1 - Tx9 per axis can be either positive (+) or negative (-). Positive values will be assumed if no sign is entered along with the register's value. These values can be entered through the CTA10-1 keypad or through the RS232 port on the front if the TRANS 01-D. Register Tx0 is used in the NC program to clear the Tool Correction memory (i.e., the X axis is T10, the Y axis is T20, etc.). External registers do not have a clear, as any new value entered will overwrite the existing value.

When the TRANS 01-D is operating, the correction value in the specified tool correction register will be added to the programmed position value. The target position of the TRANS 01-D is the sum of the programmed position and the correction value. If the total distance programmed exceeds one of the software Travel Limits, the TRANS 01-D will issue the soft fault "Axis position is out of bounds".

Zero setting of the tool correction memory is accomplished by specifying tool correction register Tx0. Tx0 is only used to clear the value in the correction memory and cannot be used as an actual tool offset value.

Tool Correction registers can be viewed in both "Manual" and "Automatic" modes by pressing the Tool Correction Register key on the CTA10-1. Tool Correction registers can only be edited in "Manual" mode.

Entering Tool Correction Register Values

The TRANS 01-D allows the tool-setter the ability to enter one or more correction values to compensate for tool wear or, when changing tools, to compensate for differences between tools. The operating NC program is written to refer to one of these values by its tool correction register number (as described on the previous section). The correction value in the referenced register is then added to the programmed dimensions to compensate for tool wear or the difference between tools.

Note: Note that tool correction values can be positive or negative, up to +/- 99.0000 inches (99.000 mm). The value entered into a register is limited by the axis parameter Aa15 "Maximum Tool Correction".

Programming Tool Corrections

To program a Tool Correction offset into a program NC Block, the function command Txx must be used. When the Txx function is programmed in a NC Block, the Tool Correction register programmed must coincide with the axis programmed (see table on previous pages). The following example shows how a Tool Correction register is specified in a program NC Block.

N001 G01 G90 G62 Z4.56 F8 **T31** M1000000

N002 G01 G90 G62 Z0.0 F150 **T30** M0100000 JS000

In this example, Tool Correction register #31 was programmed. Before this NC Block is executed, the TRANS 01-D will verify that the programmed position plus the Tool Correction register's value will not move the slide beyond a software Overtravel Limit (axis parameter Aa06). If the move will take it beyond the Overtravel Limit if executed, the TRANS 01-D will issue an error (Axis move is out of bounds) before the program NC Block is executed. It is recommended to clear the Tool Correction memory after the end of each axis' program cycle. To do this, the return NC Block in the cycle must be programmed with the Tx0 Tool Correction register for the appropriate axis. In the example above, Tool Correction register T30 clears the Tool Correction memory for the Z axis.

External Tool Correction

Tool Correction registers can also be programmed from an external device to the TRANS 01-D through the RS232/485 port on the front of the TRANS 01-D. The protocol used to do this is the same protocol used to communicate with the TRANS 01-D using "Terminal Mode". The TRANS 01-D has Tool Correction registers T01 – T03 set aside for this function. Programming these Tool Corrections is the same as it is for the internal registers.

The Tool Correction device is connected to the TRANS 01-D at the CLC/D board's serial port A (i.e., X27), using an RS-232 electrical protocol or the data can be sent via the Interbus-S PCP channel. The serial communication characteristics are as follows:

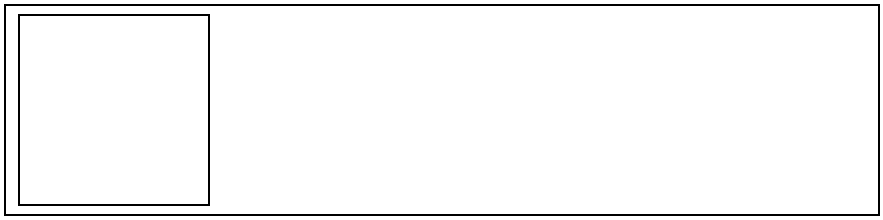
- 9600 baud
- no parity
- 8 data bits
- 1 stop bit

The Tool Correction data transmitted to the TRANS 01-D is contained in the CLC communication protocol packet described below:

```
>  FP 0.1 123...456   $cs\r\n
| | | | | | | | | | |            |    |
| | | | | | | | | | |            |    End of Message
| | | | | | | | | | |            Hex ASCII Checksum
| | | | | | | | | | Variable Length Data (Floating Point number)
| | | | | | | | | Blank (0x20)
| | | | | | | | Data Number Identifier (Floating Pt. Register #1)
| | | | | | | Period (0x2E)
| | | | | | Data Set Identifier (Active Program)
| | | | | Blank (0x20)
| | | | Command Subclass (Data)
| | | Command Class (Floating Point Variable)
| | Blank (0x20)
| Blank ((0x20)
Start Character (0x3E)
```

To compute the checksum, do a 16-bit accumulation of all of the characters before the '$'. Then add the most significant byte of the checksum to the least significant byte. Negate this value to form the two's complement. The last 2 digits of this result is the checksum to be transmitted to the CLC/D.

For example:



The response of the TRANS 01-D after receiving a valid communication packet is to simply echo back the packet header. For example, the TRANS 01-D's response to the above example would be;

```
>  FP 0.1 $1C\r\n
```

If there is an error in the data sent to the TRANS 01-D, a message is sent starting with an "!" in the data field. For example:

```
>  FP 0.1 !13 Checksum Error: 37 $2F
```

Note: The checksum of the data received by the Trans 01-D is sent back as part of the error message. In this example, the checksum is 37.

After the correct External Tool offset value has been sent to the TRANS 01-D, you must also send a handshaking signal to the TRANS 01-D to let it know that the value currently in the offset register is valid for the current NC program block. This handshaking requires you to send a non-zero value to a Global Integer. Typically a value of one (1) is sent. When the TRANS executes an NC program block with an external offset register specified, it checks to see if the Global Integer's value is not zero (0). To set this value to one (1), send the following data string to the TRANS 01-D after the correct value has been sent to the offset register:

```
>  GP 0.2 1
```

- G - indicates a global Integer.
- P - indicates you want to read/write data.
- 0.2 - requests access to Global Integer #2.

- The “1” is the data value sent. This can be any non-zero value.

When the T01, T02, or T03 is executed during program block execution, the TRANS 01-D will look for this Integer value to be non-zero. If this value is zero, the TRANS 01-D will stop NC program block execution and issue an error message to tell the user this situation exists. If the value is non-zero, the TRANS 01-D will transfer the tool correction value that has been written to float variable #1 to the corresponding offset depending on whether T01, T02, or T03 is programmed and then set the handshake global integer (GI2) to zero. Doing this prevents the TRANS 01-D from executing another cycle with an invalid data value.

The Tx0 value that should be entered into the NC program block to clear the Tool Offset is T10 for the X axis, T20 for the Y axis and T30 for the Z axis. These are the same commands used for internal Tool Offsets.

For those cases where the user wants to enter External Tool Correction values, but they do not want to implement the handshaking, the user also has the ability to send external data into the standard Tool Correction registers. In order to do this, the user must send the tool data as specified above, except the Tool Correction register specified must be one of the standard registers. When used in this way, the TRANS 01-D will execute the user program, reading the Tool Correction register’s value during NC program execution, but it will not monitor the handshaking signal for data validation.

Caution: Because there will be no handshaking, the TRANS 01-D will not set Global Integer #2’s value to 0. The user should be aware that this method does not allow the TRANS 01-D to validate the data per program cycle and could allow the wrong Tool Correction value to be used for the specific part being machined. To avoid this, the user must also make sure that the new Tool Correction value is sent down before the TRANS 01-D begins its program cycle. This will assure the user that the correct value will be used when the cycle is initiated. The checksum for the data string must still be sent with the data. When entering Tool Correction register values this way, the data header and checksum response is the only way for the external device to know if the TRANS 01-D received the data.

The table below shows the proper addressing for externally entering data into the standard Tool Correction registers.

Axis	**Standard Tool Correction register #**	**Corresponds to Floating point #**
X axis	T11 - T19	11 - 19
Y axis	T21 - T29	21 - 29
Z axis	T31 - T39	31 - 39

Syntax example: > FP 0.31 0.547 $cs

- F - indicates a floating point Integer.
- P - indicates you want to read/write data.
- 0.31 - requests access to Floating Point register #31 (standard Z axis tool correction register).
- 0.547 - data sent to standard Z axis tool correction register 31

Substituting any other Floating Point register in the above data string will send the data to the Floating Point register that corresponds to the Tool Correction register for the desired axis.

Note: The checksum must be calculated for the station address, register number, and value being sent. In the previous example, the “cs” should be replaced by the actual checksum that has been calculated.

Spindle Speed Control (NC Code S)

Spindle speeds may be programmed in any NC Block in the user program, provided it is enabled in the parameters. Spindle velocity control is achieved via a SERCOS command. Spindle speeds from 0 to the maximum spindle speed (set in parameters) may be programmed. The value is programmed directly in output speed (e.g., tool RPM). Whenever a NC Block contains a spindle speed command, the spindle will first be commanded to run at the new speed, the TRANS 01-D will wait for acknowledgment that the spindle has reached that speed then the remainder of the NC Block (positioning, homing, dwell, etc.) will be executed.

Any time a spindle speed of zero is commanded, the spindle's enable is removed (spindle disabled) after zero RPM is reached. This allows the spindle to be free to move.

Spindle Positioning Control (NC Code P)

A spindle positioning function is available, and when enabled via process parameters P06, may be programmed in any NC Block in the user program. The G01 command must be used in the program NC Block along with the Pxx.x NC program command to position the spindle. Because the TRANS 01-D considers positioning the spindle as an axis positioning command, no other axis (X, Y, or Z) can be programmed in the same NC Block when positioning the spindle. Position values from 0.0 degrees to 359.9 degrees, in 0.1 degree steps, may be programmed.

After a spindle positioning procedure, the spindles enable remains high, meaning the spindle is rigidly held in position. In cases where this is undesirable, such as automatic tool changers, simply programming a spindle speed of zero RPM after the positioning procedure (i.e. in the next NC Block) will drop the spindle enable, allowing the spindle to be freely moved.

TDA, KDA or RAC (DIAX01)

The positioning of the spindle is carried out using a drive internal positioning procedure. When the TRANS 01-D executes a spindle positioning command, it sends the programmed position to the spindle drive. After the position has been sent to the drive, the TRANS 01-D will initiate the positioning procedure. This internal procedure will position the spindle according to the parameter values set for speed and direction. Therefore, the sign of any position value programmed will be ignored. It will also send a return signal to the TRANS 01-D once the positioning is complete. Every spindle positioning command sent to the DIAX01 spindle drive will re-initiate the internal positioning procedure. Because of this, if the same position is programmed in successive blocks, each block executed will result in the spindle moving to the programmed position, even if the programmed position is the same for each block.

DDS 2.1, DDS 3.1, DDC 1.1 or DKS 1.1 (DIAX02)

When this servo drive family is used as a spindle drive, it uses two different procedures for orienting. To operate these drives as spindles, the mode of the drive must be set to "rotary". This will allow the user to position the axis within the rotation of the tool. In rotary mode, there are three possibilities for positioning direction. If shortest path is chosen, the drive will position the spindle the same as it would position any rotary axis. If Positive or Negative direction are chosen, the drive will execute the internal "Drive Controlled Homing procedure" to position the spindle, using the Marker Pulse as its reference and the programmed position as its offset and zero reference. Therefore, the sign of any position value will be ignored. Because it uses the Homing procedure, it will position

differently than when shortest path is chosen. The differences are listed below.

1. Shortest path, If the user has chosen “shortest path” for orienting, when the TRANS 01-D executes the spindle positioning command, it will move the spindle to the programmed position taking the shortest path (less than 180°) from its current position
2. Positive direction - If the user has chosen positive only for orienting, when the TRANS 01-D executes the spindle positioning command, it will move the spindle to the programmed position using the Drive Controlled Homing procedure. The speed and direction it will use for the positioning will be determined by the values entered into the axis Homing Speed and Homing Direction parameters. Because it is using the Homing procedure, once the axis is positioned, any subsequent positioning command will cause the axis to re-home itself to the new position. The exception to this is if the subsequent positioning command is to the same position, the axis will not re-position itself.
3. Negative direction - If the user has chosen negative only for orienting, when the TRANS 01-D executes the spindle positioning command, it will move the spindle to the programmed position using the Drive Controlled Homing procedure. The speed and direction it will use for the positioning will be determined by the values entered into the axis Homing Speed and Homing Direction parameters. Because it is using the Homing procedure, once the axis is positioned, any subsequent positioning command will cause the axis to re-home itself to the new position. The exception to this is if the subsequent positioning command is to the same position, the axis will not re-position itself.

DDS 2.2, DDS 3.2, DKR (DIAX03) and HDD, HDS (DIAX04)

When using either of these two drive families for spindle drives, the user must use drive firmware type SHS. Drive firmware type SSE should not be used in these drives for spindle applications. These drive families, with SHS type firmware, operate similar to the DIAX01 family, except the spindle is positioned directly with the programmed position. The direction of the orientation can be changed by changing the sign of the programmed position.

NC Program example:

N010 G01 G90 G61 X10 Y10 S100

N011 S0

N012 G01 G90 P180 F100

N013 JS000

Explanation:

N010 ***//*** Spindle Reaches 100 RPM before X & Y axes begin their motion

N011 // Spindle speed reduced to zero (0)

N012 // Spindle orientates to 180° at 100 degrees per minute

N013 // Program jumps to block N000 and stops

Auxiliary Functions (NC Code M)

Auxiliary function outputs are available in the system. They are used to operate position dependent functions such as solenoids, switches,

clamps, full depth indicators, lights that must be turned on dependent on a position, etc.

Auxiliary functions can be turned on, off or left unchanged by entering a 0, 1 or a 2 in the proper command positions. When an auxiliary function is selected, it is turned on or off at the completion of the G-code (movement or dwell).

An acknowledgment may be required for the programmed auxiliary functions, depending on the I/O system used. When the command is executed, the TRANS 01-D awaits the acknowledgments for any functions which were turned on or off before it executes the next NC Block. Once an acknowledgment is issued, the signal line must be held in that state until the function output changes. If not, NC program execution halts and a soft fault occurs.

Auxiliary Acknowledgment Definition:

M 0 1 2 1 1 1 1 1 1 0 0

0 - requires input to be always low

1 - requires input to be always high

2 - does not monitor the state of the input. i.e., if the input changes from 0 to 1, the TRANS 01-D will not issue a fault.

There is one important exception to the above. The TRANS 01-D allows a jump to the reverse program to be performed even if the acknowledgments do not match their associated auxiliary outputs. This can occur on an emergency return or upon recovery from a power failure. In this case, in the first NC Block of the Homing program it is necessary to insure that all auxiliary outputs are forced into a state where they match their acknowledgments. The easiest way to do this is to program the first NC Block of the reverse program (usually 195) to turn off all auxiliary functions and program the next NC Block with a Homing function. Assuming all acknowledgments will also be off then, homing will always be possible. When programming this, of course, it must be certain that axis movement is possible and safe with all outputs off.

NC Program example:

N100 M00010122211 JS000

Explanation:

N100 ***//*** Acknowledge state of inputs then jump and stop program on block 000

Program Jumps

Several types of program jumps are available, as discussed in the following paragraphs. If program jumps have been selected in a NC Block together with other functions, their execution will occur at the end of the NC Block, after all other functions have been executed.

Note: Block Jumps cannot be programmed in a NC Block if motion is programmed with G62 (Without Lag Finishing).

Unconditional Jump (NC Code JN)

With an unconditional jump the TRANS 01-D transfers control to another NC Block anywhere in the program. This allows the programmer to change the sequence of NC program execution. This is helpful when patching NC programs. The required new program section can be written into some available NC Block locations and tied to the original program by an unconditional jump instruction. A jump instruction at the end of the new program section can transfer control back to the original program.

Conditional Jump (NC Code JC)

A conditional jump transfers program control to the specified NC Block only if the programmed condition exists on the conditional jump control inputs. These are user interface inputs, allowing the user to determine via external signals (such as selector switch inputs) whether a programmed jump should be executed.

Example: Assume the TRANS 01-D has three different programs which start at the following NC Block numbers:

Program 1 --- Block 015
Program 2 --- Block 026
Program 3 --- Block 034

Any of these three NC programs can be executed, based on the position of a selector switch, by programming the following routine (starting at NC Block N000) with conditional jumps.

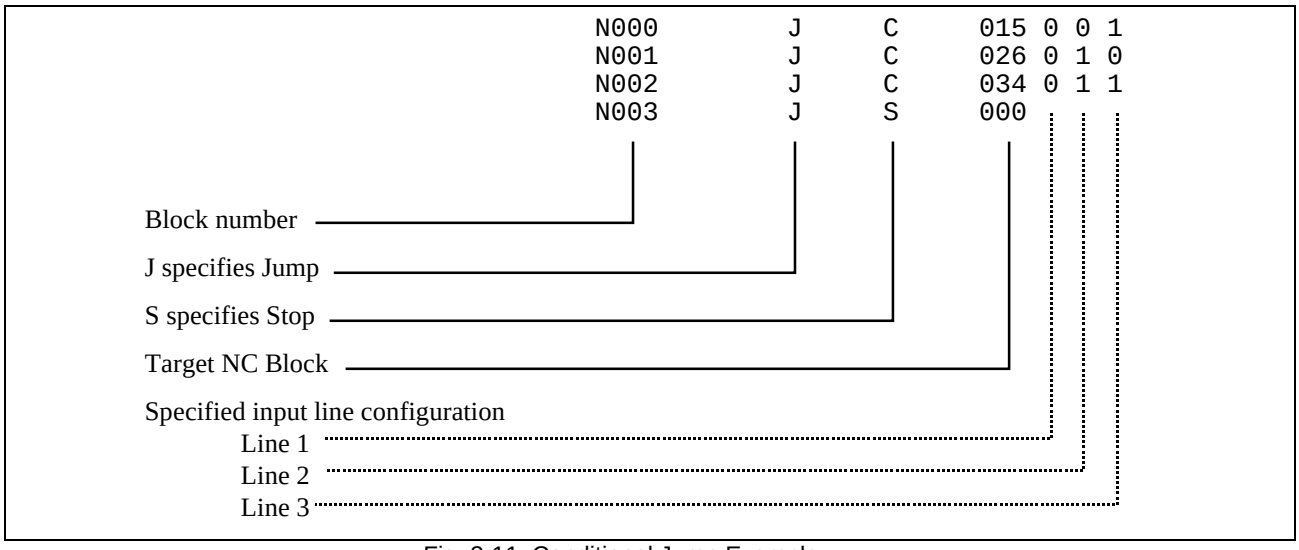


Fig. 3-11: Conditional Jump Example

In the above example, the program starting at NC Block N015 is executed when Line 3 of the selector switch is high and all others are low. The program at NC Block N026 is executed when Line 2 of the selector switch is high and all others are low; and the program at NC Block N034 is executed when Lines 3 and 2 are high and Line 1 is low.

Block 003 is programmed with a Jump To Block 000 and Stop. In the case where the selector switch has an invalid input combination, this will cause the NC program to return to the beginning and stop, rather than proceeding in an uncontrolled manner.

Note: Conditional jumps may point only to NC Blocks that contain valid program instructions.

Jump And Stop (NC Code JS)

The Jump And Stop instruction causes an unconditional jump to the target NC Block and subsequent stop of the program without execution of that target NC Block. Continuation of the NC program occurs only after a Start signal is issued again.

This instruction is used mainly at the end of a machining program, where a Jump To Block 000 And Stop instruction is required. Jump and Stop can also be used at other positions in an NC program, if it is desirable to continue the NC program only after a renewed state.

Reverse Vector (NC Code JR)

When an executing user program is interrupted with a Home (Reverse) signal, and the axis is to be returned to the Home position, it is often necessary to execute different NC program sequences depending on the status of the user program at the time the Home (Reverse) signal was received. For example, if the tool is in the part, your Home (Reverse) program may be different than if the tool were at the face of the workpiece. This situation may occur in slide units, both during manual operation, when the Reverse input is triggered, and during automatic operation when the Homing input is triggered.

A special type of jump command, the Reverse Vector Jump, specifies with which NC Block the Return program is to start when a Reverse or Homing signal is issued. In the original TRANS01 "Blue Box" and TRANS 01-D version 5, the reverse vector was set to NC Block N195 when the TRANS 01-D was reset. However, in the TRANS 01-D version 6, the reverse vector is maintained as the last programmed reverse vector. It is recommended that block N195 be programmed with the basic homing program. At any point in the forward program you can use the Reverse Vector Jump command to set some NC Block other than NC Block 195 as the start of your Return (Reverse) program.

A starting point programmed in this manner will remain effective until it is replaced by a new reverse vector of the same type executed in your NC program. This allows coordination of very complicated Reverse programs within the user program with a minimum of programming overhead.

Reverse Vector JR000

Because NC Block 000, the starting NC Block of the program, could never be used as a reverse vector, reverse vector JR000 is used for a special purpose. Whenever the TRANS 01-D executes a NC Block containing a reverse vector of JR000, it considers all following NC Blocks to be the return portion of the NC part program, even if they specify forward motions. Reverse vector JR000 is useful primarily in “Manual” mode, but also has an important effect in “Automatic” mode.

In “Manual” Mode -- You should program an NC Block containing a Reverse Vector Jump to JR000 to indicate the end of the Forward program (profile). In “Manual” mode, pressing and holding the FORWARD button at the operator Station causes the TRANS 01-D to execute the Forward program. When the TRANS 01-D completes execution of a NC Block containing a reverse vector of JR000, the Forward input will be ignored and only the Return (Reverse) input can be used.

Note: if reverse vector JR000 is not programmed at the end of the Forward program, depressing the Forward button while in “Manual” mode will cause the TRANS 01-D to execute the entire NC program, both the Forward and Reverse profiles.

It is good practice to program a homing command in the NC part program after reverse vector JR000, however it is not required. The TRANS 01-D operates correctly without the homing command in all cases, except where all power has dropped after execution of reverse vector JR000 and before execution as the Jump to Block 000 and Stop.

Some users omit the homing command in the reverse vector program to reduce cycle time. Instead, they program an absolute move to the home position. Unlike many other controls, the TRANS 01-D executes a complete homing process only when first powered up if an incremental encoder is present. If the system contains all multi-turn encoders no referencing is required. Subsequent homing commands are essentially an absolute move to the home position, with the TRANS 01-D remembering where Home is and checking that it is reached when commanded.

’Jump on Event’ Program

Jump on Event is a jump that is executed when the Jump on Event input to the TRANS 01-D goes high. When the Jump on Event input goes high, the TRANS 01-D immediately jumps to NC Block N170 and begins to execute the program from that point. When this happens, the TRANS 01-D sets an internal flag to specify that it is running a “Jump on Event” program. While this flag is set, no other Jump on Event inputs are recognized. After the “Jump on Event” program is finished, the flag is reset.

Programming Blocks for the ’Jump on Event’

The 'Jump on Event' program begins with NC Block N170 and ends with program NC Block N179. The 'Jump on Event' program can be extended if a jump is made to another section of programming NC Blocks that are available other than N170 to N179.

Programming Procedure

Inside a 'Jump on Event' program, you can use all the same programming options as in the normal TRANS 01-D program. The program must start in NC Block N170 and must end either with a programmed jump in NC Block N179, with a 'Jump and Stop to 000', or a JReturn, which causes the TRANS 01-D to return to the NC program block it was executing when the Jump on Event input went high.

```
N000  G01   X99   F500
N001  G04   F2    S1000
N002  G01   X110  F500

N003 G01 A X50 F500 Traverse
N004 JS000
```

```
Signal JUMP ON EVENT

N170 G01 G91 X10 F50
N171 JU070
N172 JN 179
N179 JN 003

N070 G04 F2 S500
N071 G01 G90 X99 F100
N072 Jreturn

’JUMP ON EVENT’
Program
```

Fig. 3-12: Jump On Event Programming Example and Program Sequence

Jump to Subroutine JU

NC programs which contain identical program sequences in two or more places can be simplified by designating these identical sections as subroutines or subprograms. These subroutines can be executed (called) from the main program by executing a Jump To Subroutine instruction.

When the Jump To Subroutine occurs, the TRANS 01-D transfers program execution to the NC Block number specified in the Jump instruction while storing the number of the NC Block which initiated the jump to subroutine. When the TRANS 01-D encounters a JReturn instruction in the subroutine sequence, it returns program control to the main program NC Block from which it executed the Jump To Subroutine instruction. NC program execution then continues with the next NC Block. A Jump To Subroutine can be executed from any point in the main program and a return to that point is assured after completion of the subroutine.

Note: A Jump To Subroutine must jump to a valid NC Block.

Subroutines are programmed just like main programs. Note, however, that a subroutine must always contain a JReturn instruction as its last entry. However, a JReturn may not appear in a NC program that has not

been declared as a subroutine, because this would confuse the program sequence.

JReturn

Used to return the user program back to the block that was left with a JU (jump to Subroutine.) This block jump can also be used to return the program back to the block that was being executed when the Jump on Event input was set high.

4 CLC DRIVE PARAMETER EDITOR

4.1 Overview

When opened, the **CLC Drive Parameter Editor** uploads the current status information for drive #1 (the default). The current drive status is also displayed. The position, Velocity and Acceleration values from the CLC / TRANS 01-D card are displayed along with the feedback status from the selected drive.

The *Drives* box allows selection of another drive by entering a drive number or scrolling with the up/down list buttons. Clicking ***Update*** uploads the current values for the selected drive.

Since the drive internally generates rate profiles for single axis motion, the programmed acceleration is also displayed. Acceleration is not shown for coordinated motion since the CLC / TRANS 01-D path planner manages acceleration for coordinated motion.

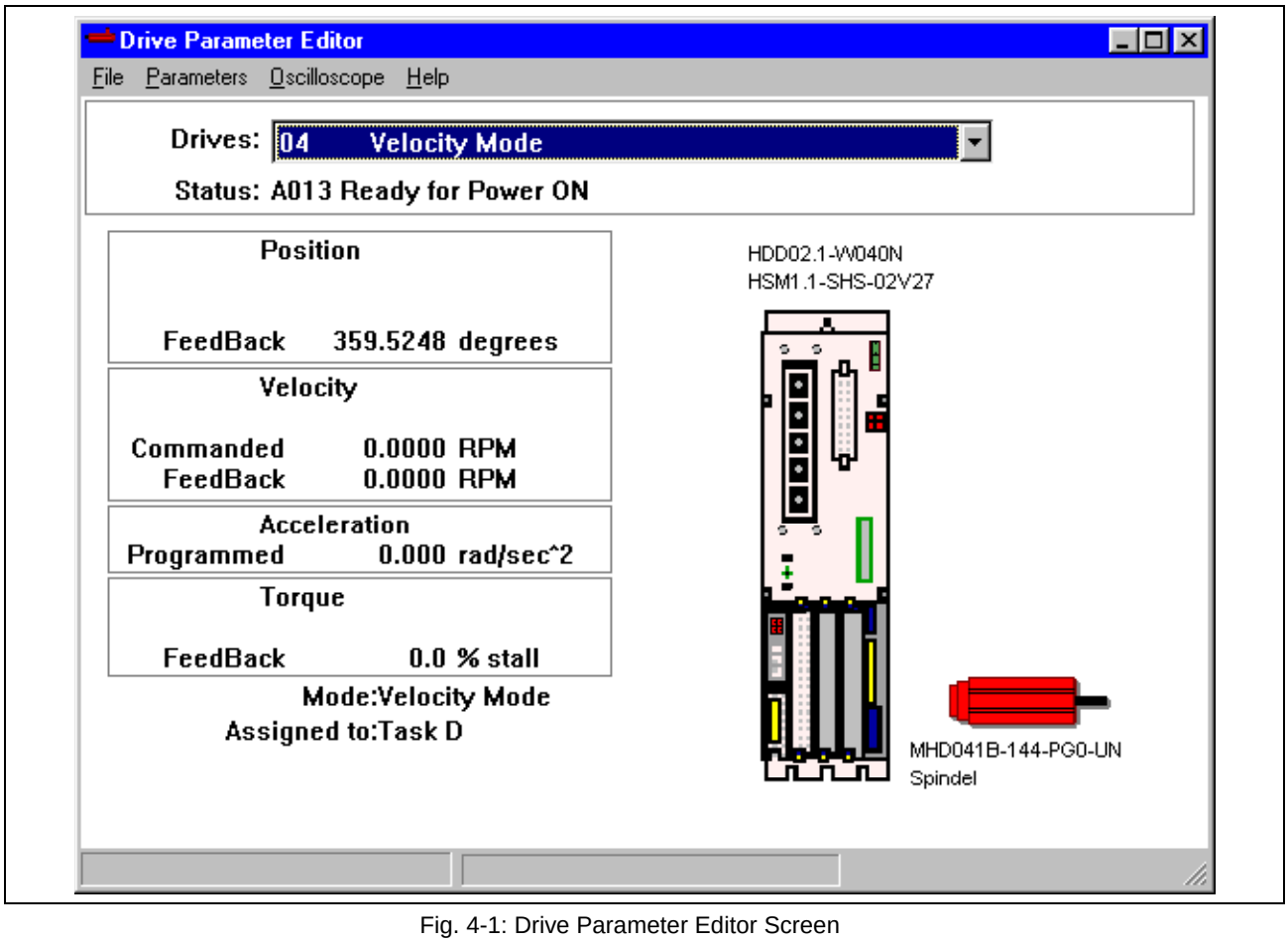


Fig. 4-1: Drive Parameter Editor Screen

4.2 Parameters Menu

Analog Outputs

Choosing **Analog Outputs** from the **Parameters** menu opens the Drive "n" **Analog Output Editor** dialog box. This is used to manage signal

selection and scaling of the AK1 and AK2 analog output drive signals. By default, the current values for drive #1 are uploaded and displayed.

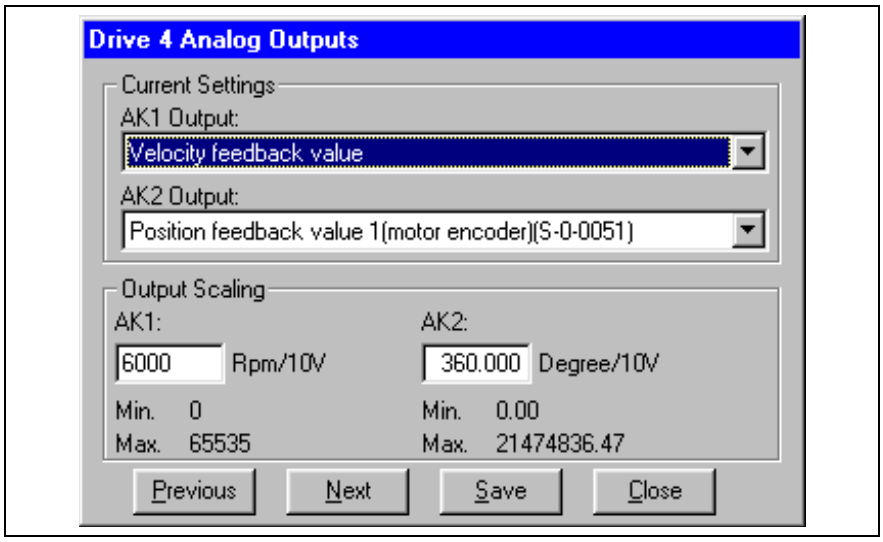


Fig. 4-2: Analog Outputs Screen

The Drive # box allows selection of another drive by entering a drive number or scrolling with the up/down list buttons. Clicking ***Update*** uploads the current values for the selected drive. Scrolling lists of signals available for both outputs include:

- No output
- Current command value
- Velocity feedback value
- Velocity command value
- Position command value difference between each SERCOS cycle (2ms).
- Position feedback value 1 (motor encoder)
- Position feedback value 2 (external encoder)
- Lag error
- Sine signal from motor encoder
- Cosine signal from motor encoder
- Sine signal from external encoder
- Cosine signal from external encoder

Individual data entry boxes allow independent scaling of each output (maximum 10 volt output) to meet the requirements of an external indicator (analog or digital panel meter, etc.)

Clicking on the ***Save*** button downloads the signal selections and scaling factors to the selected drive through the CLC / TRANS 01-D and SERCOS communications.

Drive Direction

The **Drive Direction Editor** is an edit window for viewing and setting drive direction parameters. These parameters invert the direction of the commands to the drive (i.e., A 5 inch move will move 5 inches negative if the Position Command is set to the reverse direction.)

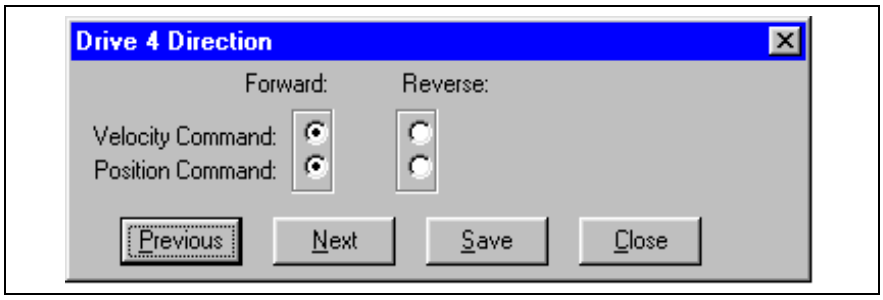


Fig. 4-3: Drive Direction Screen

To view settings for a drive, select the drive number in the main window and press the ***Update*** button.

To change a drives settings, switch to parameter mode, then select the direction buttons and press the ***Save*** button. ***See drive parameters S-0-0043 and S-0-0055***.

Drive Name

This window displays the name of the currently active drive. The default name is the axis number however a custom name can be assigned and saved to any of the drives on the SERCOS ring.

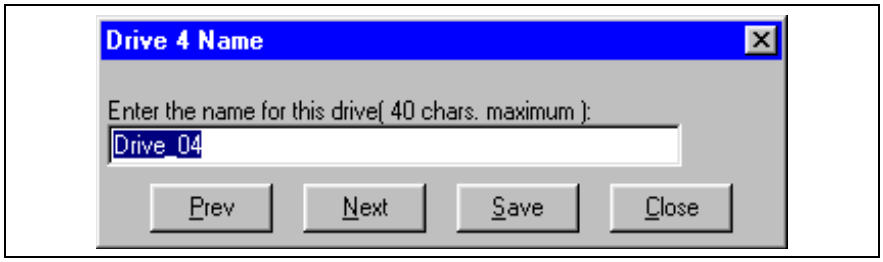


Fig. 4-4: Drive Name Screen

Drive Monitoring

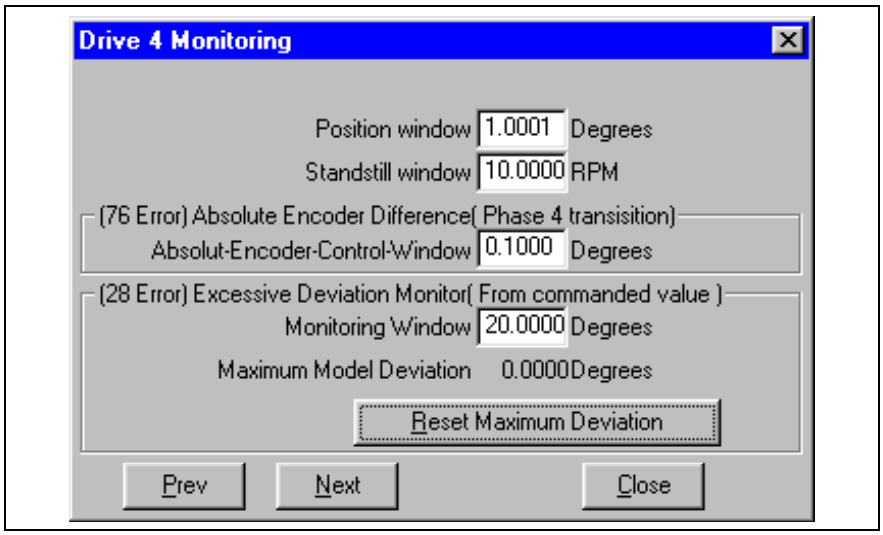


Fig. 4-5: Drive Monitoring Screen

Selecting Drive **Monitoring** from the **Parameters** menu opens the Drive **Monitoring** window and refreshes the following drive information:

Position Window (S-0-0057) sets the tolerance distance used to determine if it's in position.

Zero Velocity Window (S-0-0124) sets the velocity to determine if it's at a standstill.

Absolute Encoder Difference (P-0-0097) sets the maximum distance the motor can move when off, without causing an error 76 on power-up to phase 4.

Monitoring Window (S-0-0159) sets the maximum position unit (or percent for some drives) from the command value before the drive issues an error 28.

The current *Maximum Model Deviation* from the command value is displayed in position units (or a percent for some drives). This value can be reset using the button on the bottom of the screen.

Drive Tuning

Selecting **Drive Tuning** from the **Parameters** menu opens the **Drive Tuning** dialog box for the currently active drive. The adjustments within the *Velocity Loop* are related to a machines performance. The *Current Loop* adjustments are set according to the respective motor/drive combinations and should not be altered from their initial operative settings.

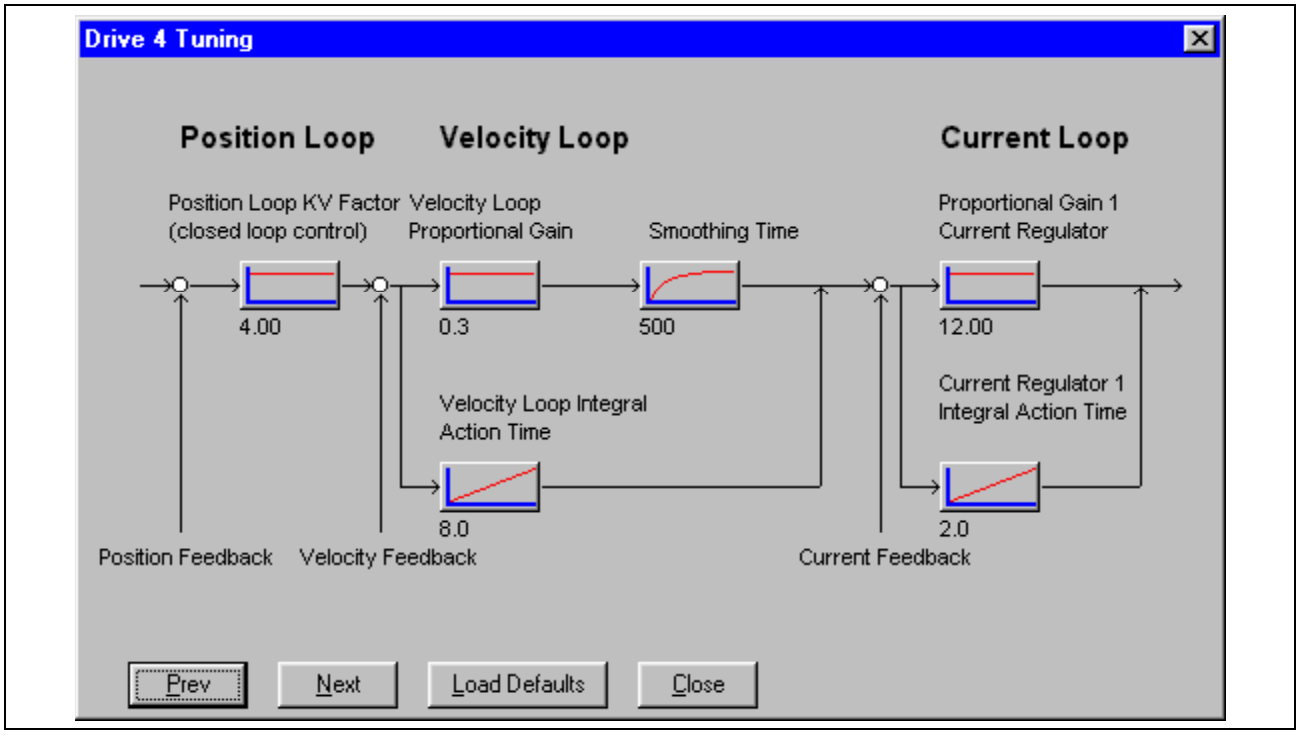


Fig. 4-6: Drive Tuning Screen

The *Velocity Loop Proportional Gain* data entry box adjusts the gain of the loop feedback path (SERCOS parameter S-0100). The gain is initially adjusted by the drive/motor combination for a 1:1 load/motor inertia ratio. This parameter value may be set from 0 to 65535

The *Smoothing Time Constant* data entry box sets a low-pass filter that limits the bandwidth of the feedback loop and reduces digital quantization effects (Indramat parameter P-0-0004). The time constant is set in microseconds, any entry under 250µs switches off filtering.

The *Velocity Loop Integral Action Time* data entry box also sets a low pass filter time constant integrating the velocity loop feedback signal (SERCOS parameter S-0-0101). This parameter is typically used to adjust the loop response time, matching the load to motor and reducing overshoot that may result from a rapid (step) change. This filter has a lower frequency breakpoint than the Smoothing/Roll-off value.

The *Proportional Gain 1 Current Regulator* (S-0-0106) and *Current Regulator 1 Integral Action Time* (S-0-0107) adjustments are used for the initial tuning of respective motor/drive combinations. These adjustments should not be changed once they are set for a specific system.

Clicking the Defaults button loads default parameter values into the dialog box data entry fields (CLC / TRANS 01-D must be in Parameter Mode). The default values assume a 1:1 ratio of load inertia to motor inertia.

For further information on tuning Indramat drives and tuning parameters refer to the ***Digital Servo Drive User's Manual.***

Drive Limits

Positive and Negative *Travel Limits* (i.e., *Software Position Limits*) set floating point values for the drive Positive (S-0-0049) and Negative Position Limit Value (S-0-0050) parameters. The *Travel Limits* (i.e., *Software Position Limits*) are not affected unless *Enabled* is checked.

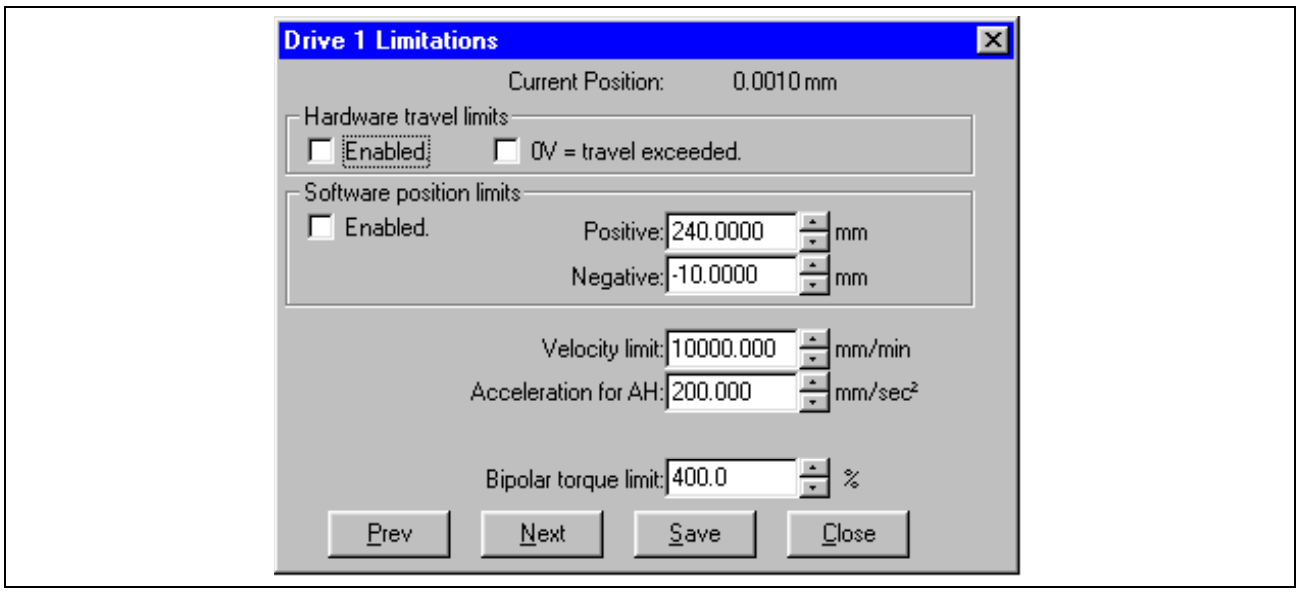


Fig. 4-7: Drive Limits Screen

The *Velocity limit* value (S-0-0091) determines the maximum allowable speed in either direction. If the velocity limit value is exceeded, the drive responds by setting the message "ncommand > nlimit" in Class 3 Diagnostics (IDN S-0-0013).

The *Acceleration for AH* parameter (S-0-0138) reduces the maximum acceleration ability of the drive symmetrically around 0, to the programmed value in both directions.

The *Bipolar torque limit* value (S-0-0092) determines the maximum allowable torque in either direction. If the torque limit value is exceeded, the drive sets the message "T > Tlimit" in Class 3 Diagnostics (S-0-0013).

Note: The ***Save*** button and all of the value fields will be grayed out if the system is not in *parameter mode*.

Drive Reference

This menu item will automatically sense the active drives motor encoder type and launch either the single or multi-turn encoder dialog box.

Single-turn Encoder

Selecting **Drive Reference** from the **Parameters** menu opens the **Single Turn Encoder Homing Setup** dialog box and uploads the current parameter values for drive #1 (the default).

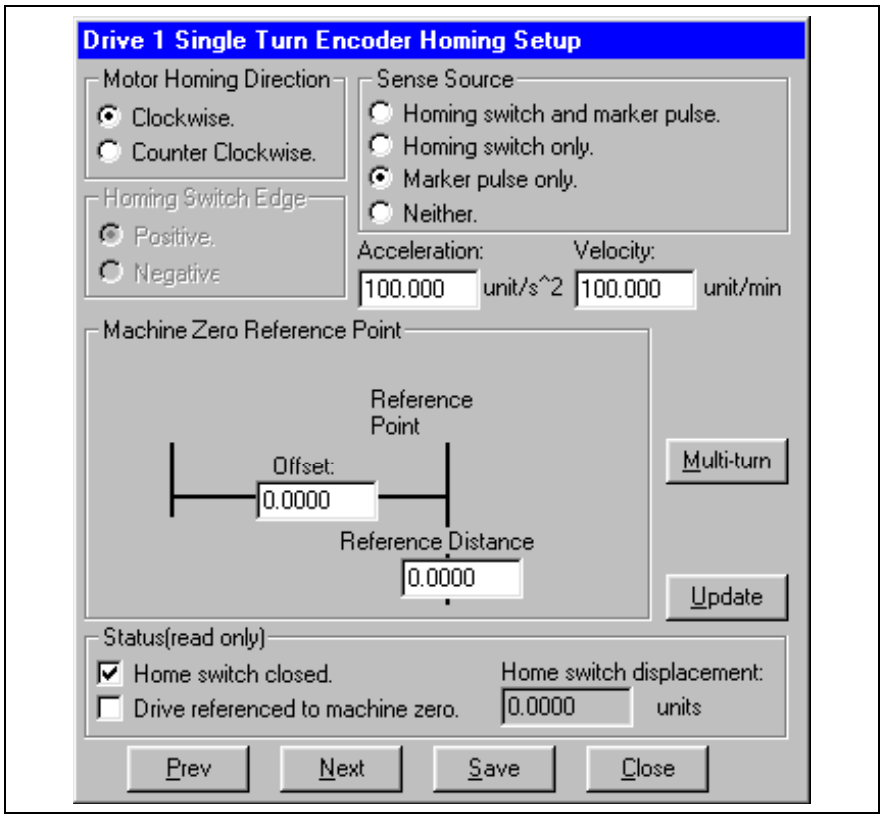


Fig. 4-8: Drive Reference - Single Turn Encoder Screen

Homing does not need to be configured if the Home icon or AXIS/HOME text language command is not used in user programs. Homing is an internal capability of Indramat intelligent drives and requires only that the CLC / TRANS 01-D send a home command to the drive. The actual homing procedure performed by the drive is set by the drive parameters.

The *Drive #* box allows selection of another drive by entering a drive number or scrolling with the up/down list buttons. Clicking ***Update*** uploads the current values for the selected drive.

Because the setup of drive homing parameters is highly system dependent, refer to the ***Drive User's Manual*** for information on the various homing routines and the requirements for setting up the homing parameters.

Multi-turn Encoder

The **Multi-turn Encoder Homing Reference** instructs the DDS drive to use the current shaft position as the position indicated in the *'Reference Distance'*.

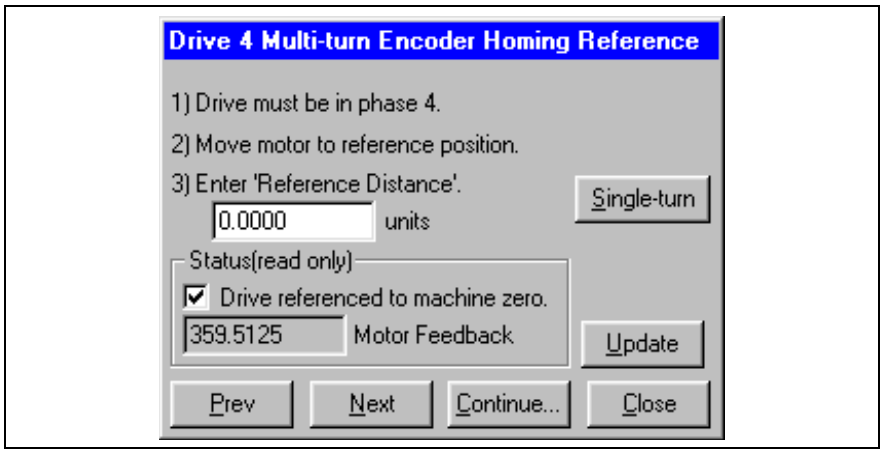


Fig. 4-9: Drive Reference - Multi-turn Encoder Screen

Set the drive number to the drive to be referenced. Move the motor to the position to be referenced. Set the *'Reference Distance'* to the value to be assigned to this position.

Press the ***Continue...*** button. A second screen will inform you that all motion on this drive will be referenced to these values. Click on ***OK*** to accept, or ***Cancel*** to abort.

After successful completion of this procedure, the checkbox *Drive referenced to machine zero* will be checked.

Encoder 2

The Encoder 2 setup displays three consecutive windows that are used for configuring an external encoder for each drive in a system. Before an external encoder can be setup, the system must placed in parameter mode. Each drive having an external encoder requiring setup must first be selected from the drop-down list in the Drive Parameter Editor main window.

Note: The three Encoder 2 Setup windows contain a *Next* and *Previous* button for moving between the setup windows and not for selecting the next drive as used in other VisualMotion windows.

Select one the following **Function** types in Fig. 4-10 and click on the *Save* button before proceeding to the next screen. The available function types are as follows:

- Additional load side feedback
- ELS Master
- Single load side feedback
- Measuring wheel (Not supported with TRANS01-D firmware)
- Spindle

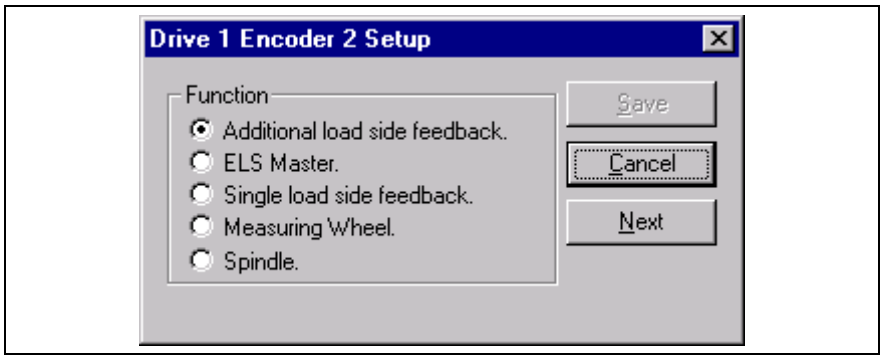


Fig. 4-10: Drive n Encoder 2 Setup

Clicking on the *Next* button will display the second setup window where the Encoder Direction, Application Type and Encoder Type are selected.

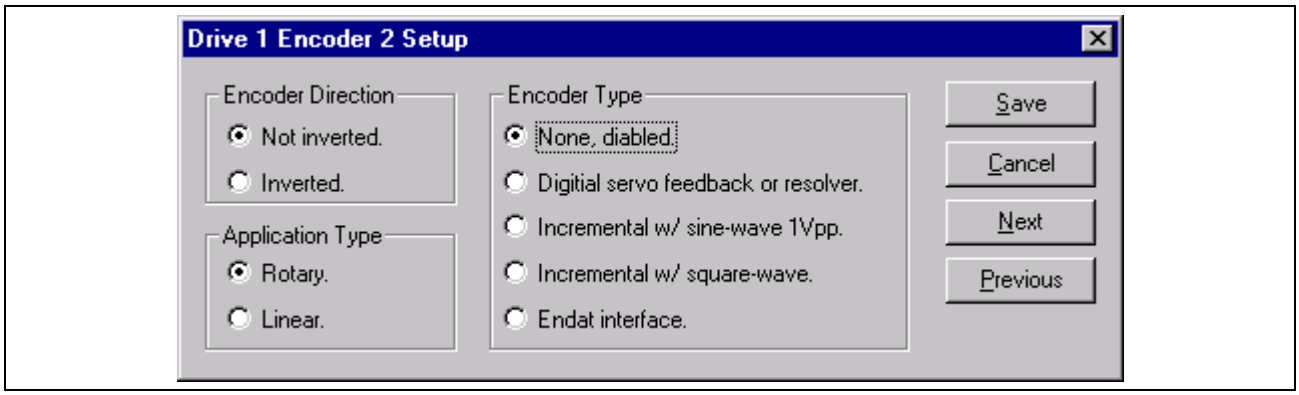


Fig. 4-11: Drive n Encoder 2 Setup (second window)

Encoder Direction sets the rotational direction of the external encoder with no relationship to the motor that is attached to the same drive. The rotational direction is viewed while facing the encoder's shaft.

- ***Not inverted*** is a clockwise rotation.
- ***Inverted*** is a counter clockwise rotation.

Application Type identifies the application as either rotary or linear.

Encoder Type identifies the style of the external encoder being used.

After the selections are made, click on the *Save* button before continuing to the *Next* window. Clicking on the *Previous* button will backup one window.

The last setup window in Fig. 4-12 is used for setting feedback resolution, filter time constant and monitoring window.

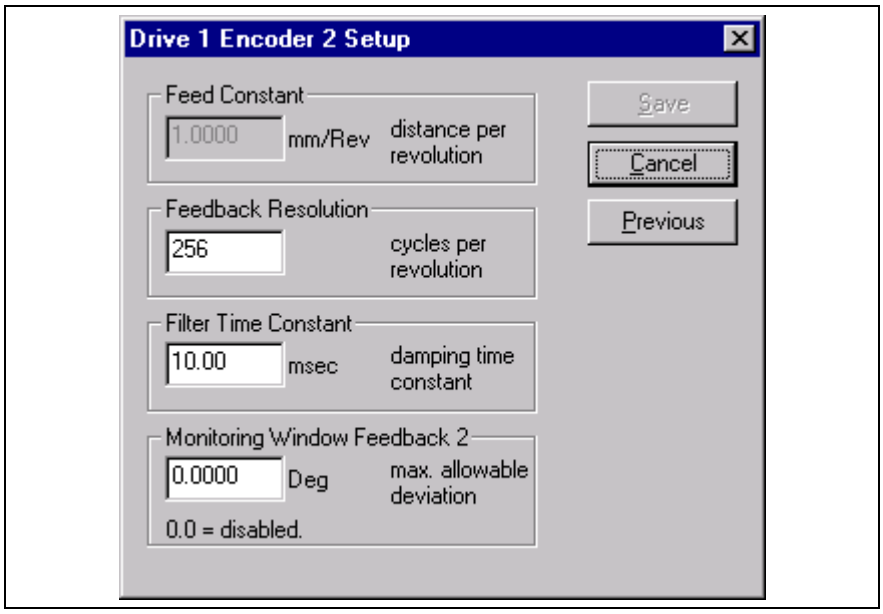


Fig. 4-12: Drive n Encoder 2 Setup (last window)

Mechanical

Selecting **Mechanical** from the **Parameters** menu opens the **Mechanical System** dialog box and uploads the current values from the drive. This dialog box allows easy access to several important parameters, which must be set before running any motion programs.

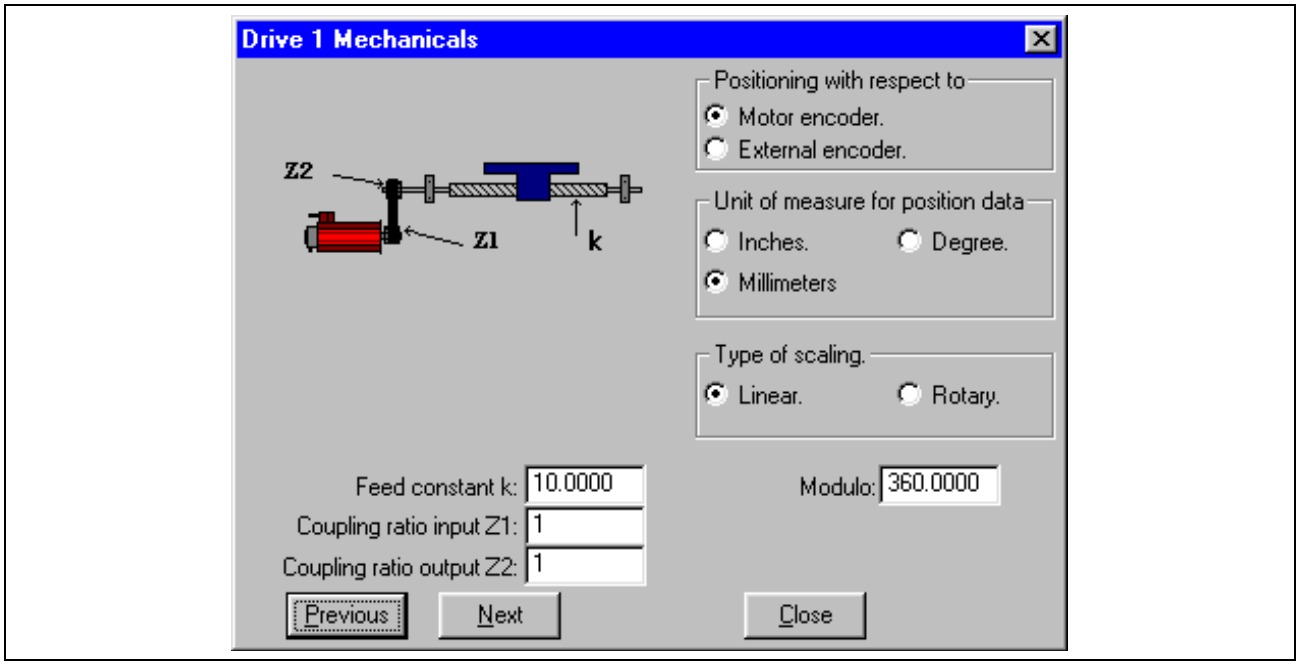


Fig. 4-13: Mechanical Screen

Data *Positioning with respect to* the load configures the drive to use Encoder 2 to close the position loop and provide cyclic feedback from drive parameter S-0-0053. This sets *S-0-0004 - Bit 11: Use Secondary Encoder for Positioning* equal to 1. This option must be set if the CLC / TRANS 01-D is used with DDS-2.1 linear motor firmware.

The *Units of measure for position data* selection has a choice of *Inches*, *Millimeters*, or *Radians* [*Degree*] for the system-wide unit of measurement (A-0-0005).

The *Type of scaling* selection can be linear or rotary (A-0-0004 bit 2). When linear is selected absolute positioning is enabled in the drive. When rotary is selected, position is in degrees, velocity in RPM, and acceleration in radians/sec2.

The *Feed constant k* allows setting the ratio of movement in system units resulting from each revolution of the driven shaft (S-0-00123). For example, a 5 TPI ball screw provides 0.200 inch movement per revolution.

Coupling ratio output Z2 and *Coupling ratio input Z1* permits setting the ratio between the motor shaft and driven shaft. Integer values permit preservation of maximum system accuracy with ratios that result in repeating decimals (i.e., 1:3 = 0.333333...). These values are set in the drive's *Input Revolutions of Load Gear* (S-0-0121) and *Output Revolutions of Load Gear* (S-0-0122) parameters.

The *Modulo* value (S-0-0103) is indicated as a maximum rotational value in which the motor will turn before resetting the position to zero. The default value when operating in modulo mode is 360. (Modulo mode is set in the drive by setting bit 7 of IDN S-0-0076, Scaling Options for Position Data).

Overview

Selecting **Overview** from the **Parameters** menu opens a **View Drive # SERCOS Parameters** dialog box. This dialog box may be used to view and setup all Drive and CLC / TRANS 01-D user accessible parameters.

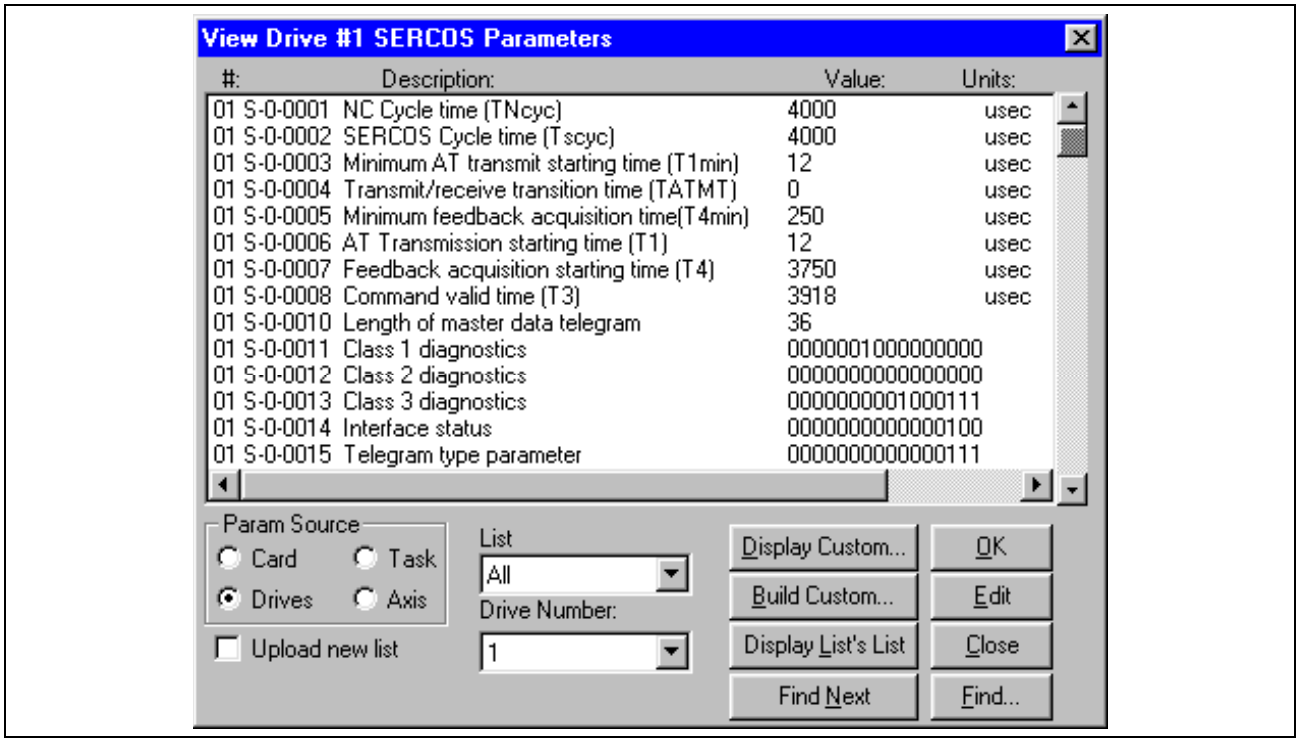

View Drive #1 SERCOS Parameters
#: Description: Value: Units:
01 S-0-0001 NC Cycle time (TNcyc) 4000 usec
01 S-0-0002 SERCOS Cycle time (Tscyc) 4000 usec
01 S-0-0003 Minimum AT transmit starting time (T1min) 12 usec
01 S-0-0004 Transmit/receive transition time (TATMT) 0 usec
01 S-0-0005 Minimum feedback acquisition time(T4min) 250 usec
01 S-0-0006 AT Transmission starting time (T1) 12 usec
01 S-0-0007 Feedback acquisition starting time (T4) 3750 usec
01 S-0-0008 Command valid time (T3) 3918 usec
01 S-0-0010 Length of master data telegram 36
01 S-0-0011 Class 1 diagnostics 0000001000000000
01 S-0-0012 Class 2 diagnostics 0000000000000000
01 S-0-0013 Class 3 diagnostics 0000000001000111
01 S-0-0014 Interface status 0000000000000100
01 S-0-0015 Telegram type parameter 0000000000000111
Param Source
Card
Task
Drives
Axis
Upload new list
List
All
Drive Number:
1
Display Custom...
Build Custom...
Display List's List
Find Next
OK
Edit
Close
Find...

Fig. 4-14: Overview Screen

Parameters may be uploaded for display or editing by selecting one of the *Param Source* radio buttons, then clicking the ***OK*** button. Choosing *Drives* or *Axis* requires the entry of a *Drive Number* or *Axis Number* in the enabled data entry box. Choosing CLC / TRANS 01-D card allows uploading the CLC / TRANS 01-D card system parameters. Selecting Task requires choosing of one of the four CLC / TRANS 01-D tasks from the enabled pull-down menu.

After the selected list has been loaded, you may scroll the list by clicking on the list scroll bars up and down arrowheads, or clicking and dragging the scroll button. Clicking and holding (or dragging) the scroll bar button will display the number of the first viewable parameter in the display list in the upper left of the title bar.

Selecting a list entry and clicking the ***Edit*** button or double clicking on the list entry opens an **Edit &&&& Parameter** dialog box that allows you to change the parameter value and update the parameter by downloading it to the CLC / TRANS 01-D or the drive(s).

In addition to uploading all the parameters of a set (i.e., all drive or task parameters, etc.) you may build a custom set of parameters by clicking on the ***Build Custom...*** button and opening a **Build Custom Display** dialog box.

When the **Build Custom Display** dialog box opens, the parent **View &&&& Parameters** dialog box remains active. Selecting a list item from the **View &&&& Parameters** dialog box list, then clicking ***Add*** in the *Parameter Numbers* box adds the selected parameter to the **Build Custom Display** list. Only one parameter may be added at one time. Selecting one of the parameters from the *Parameter Numbers* box, and then clicking ***Delete*** removes the item from the **Build Custom Display** list.

Since the parent **View &&&& Parameters** dialog box remains active, the parameter list in the parent **View &&&& Parameters** dialog box may be changed by re-selecting a different *Param Source*. Parameters from the newly uploaded list may then be added to the **Build Custom Display** list in the *Parameter Numbers* box. In this manner a custom list may be built by selecting any available parameter from any parameter list.

A previously saved custom list may be loaded and used as a source for building another custom list, however, it must be the first list loaded. You cannot load a custom display list once another list has been loaded without closing the **Build Custom Display** dialog box. Clicking on ***Save*** pops up a **Name Custom Display** dialog box permitting entry of an eight character maximum filename.

Clicking on the ***Display Custom…*** button opens a **Display Custom List** dialog box with a list of the currently saved custom parameter lists by name. Selecting a list and clicking on ***OK*** or double clicking the list item uploads and displays the parameters in the selected list.

The ***Find…*** and ***Find Next*** buttons can be used to search the *Description* section of the **View &&&& Parameters** dialog box for all occurrences of a specific string of characters.

4.3 Oscilloscope

The oscilloscope utility is used to capture and display run-time data. The capture can be of the CLC-D control or on a drive that supports this feature. Selected data is acquired on the drive or control, passed to Visual TRANS, and displayed on the graphical format. The graphical display and supporting data can be printed, or the data can be saved to a file for later review.

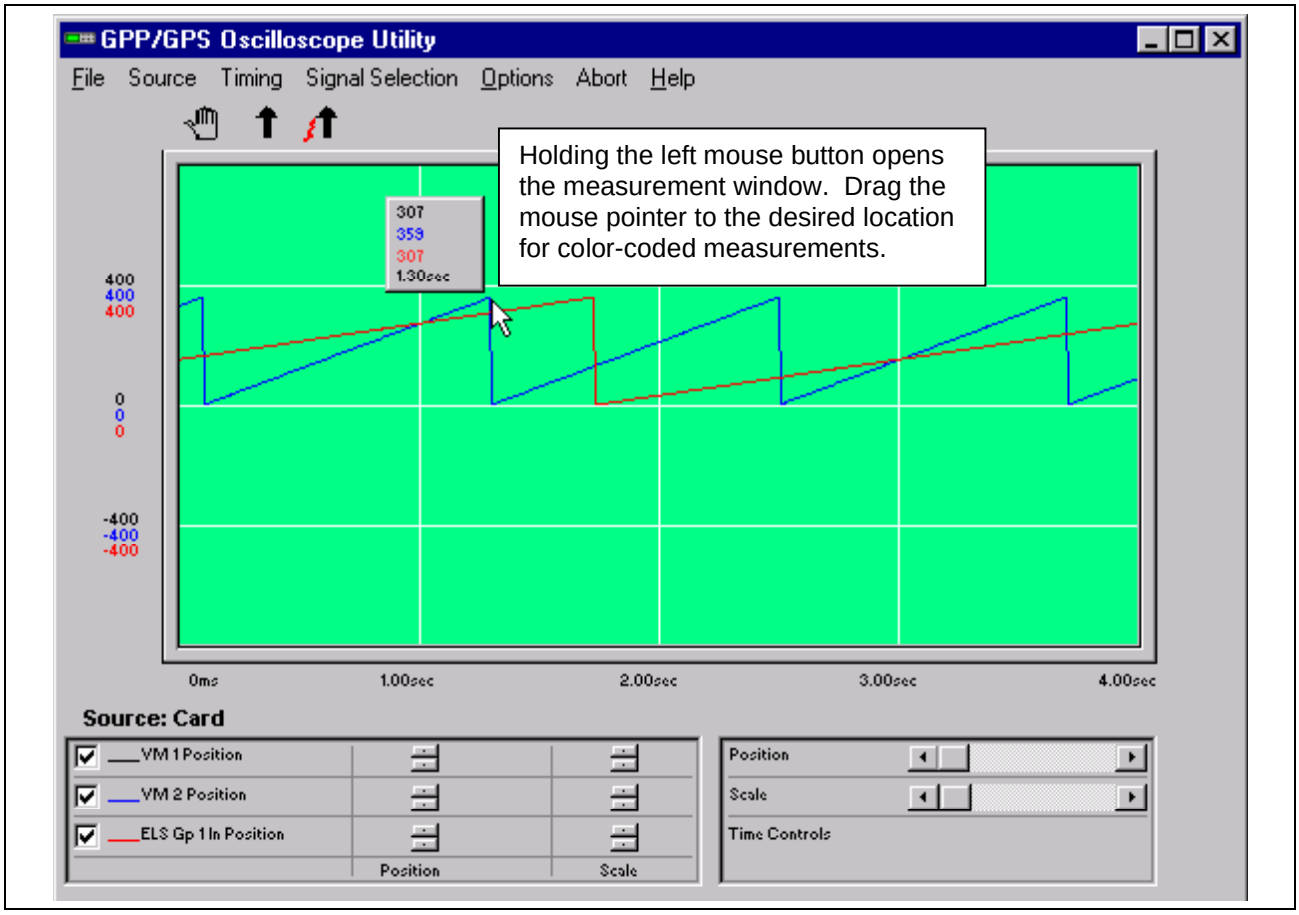


Fig. 4-15: Oscilloscope Utility

File Menu

The File menu is used for retrieving file data, saving data to a file, printing, and exiting the oscilloscope utility.

Open - data from user selected input file is loaded into input data list-box.

Save - data from user selected output is loaded into output data list-box.

Print… - the oscilloscope graph and its related data table is sent to the printer.

Exit - terminates this utility

Source Menu

Selects the source from which the oscilloscope will gather signal data.

Drive 1 to n - lists of all drives on the SERCOS ring that supports the oscilloscope feature.

Card - when selected, the oscilloscope can then read card variables, parameters and registers.

Timing

The **Oscilloscope** [timing] **Options** in Fig. 4-16 are used for setting the *Sampling Rate* (How often a trace is captured) and *Sample Count* (How many sampling rates are captured). The *Capture Duration* field displays the total capture duration that is calculated by multiplying the *Sample Count* and *Sampling Rate*. A *Pretrigger* can be added and is a percentage of the capture interval.

Note: The *Pretrigger* appears on the oscilloscope screen as a vertical line.

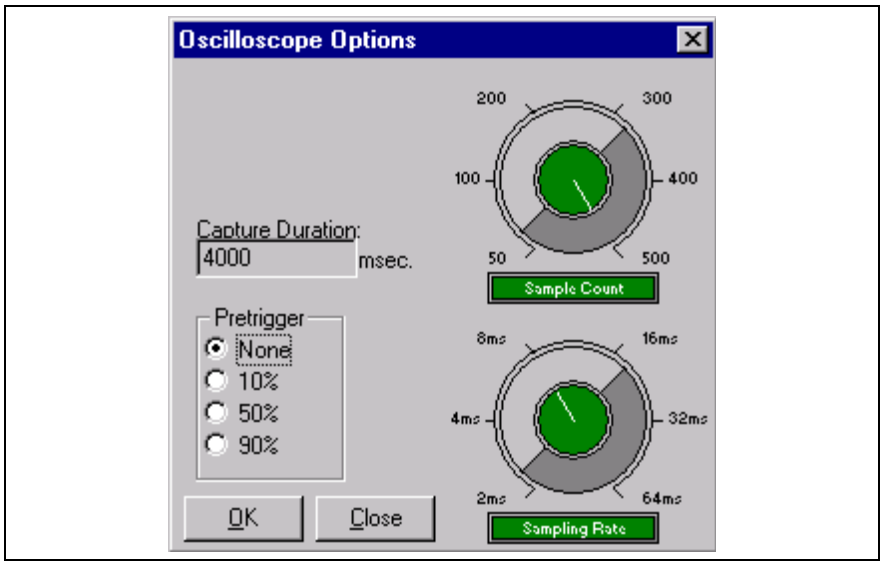


Fig. 4-16: Oscilloscope Options

Signal Selection

The **Drive Signal Setup** in Fig. 4-17 is available when a drive is selected under the **Source** menu.

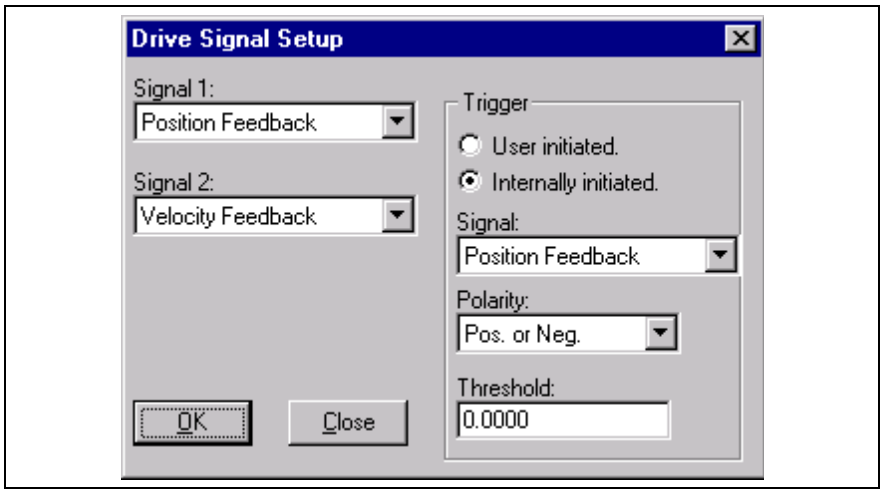


Fig. 4-17: Drive Signal Selection

Two drive signals can be captured and viewed. The following is a listing of the available drive signals.

- Position Feedback.

- Velocity Feedback.
- Velocity Deviation (from commanded value).
- Position Deviation (from commanded value).
- Torque Command Value (required to maintain the commanded Velocity/Position).
- Disabled (Signal 2 only).

The **Card Signal Setup** in Fig. 4-18 is available when **Card** is selected under the **Source** menu.

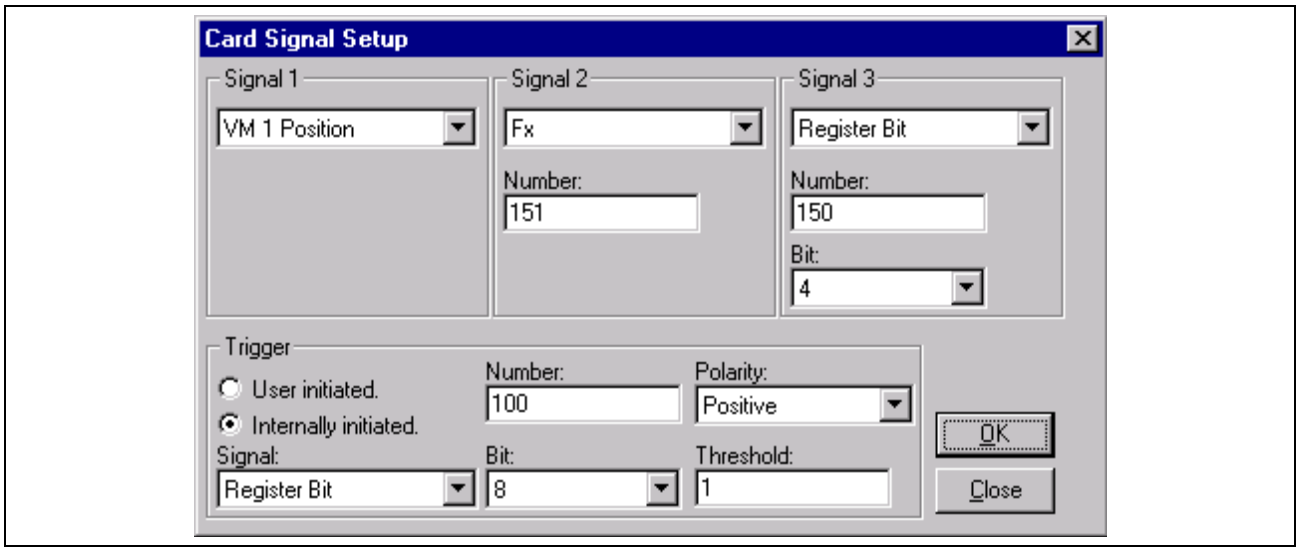


Fig. 4-18: Card Signal Selection

- Program Floats (Fx).
- Program Integer (Ix).
- Global Floats (GFx).
- Global Integers (GIx).
- Axis Parameters of drives on SERCOS ring*.
- Register Bit.
- +/- Register (could be used to monitor a register's value).
- Card Param.
- ELS Gp # In Velocity.**
- ELS Gp # Out Position.**
- VM1 Position (Virtual master signal).
- VM1 Velocity.
- VM2 Position.
- VM2 Velocity.

*Axis parameter must be in cyclic telegram. Use parameter A-0-0185 and A-0-0195 to add other drive parameters to cyclic data.

**The # symbol represents ELS Groups 1-8. This same signal is available for each ELS Group in the system.

For either signal source, the sample acquisition may be *User initiated* or *Internally initiated*.

For *User initiated* captures, data acquisition starts as soon as the capture button is pressed. This type of start capture is not deterministic. All other fields in the *Trigger* section are grayed out.

For *Internally initiated* captures, the available signals are the same as the signals for *Signals 1 – 3*. The heading in the *Trigger* fields will change based on the signal selected. The trigger *Polarity* options are on "*Positive*" edge, "*Negative*" edge, or both (i.e., "*Pos. or Neg.*"). The signal *Threshold* is the signal level to trigger.

Options menu

From the options menu, a trace's appearance can be changed from **Lines** to **Dots**. When combined with the *Time Controls* feature in Fig. 4-27, the user can scale (zoom) in to reveal the individual dots that makeup the trace.

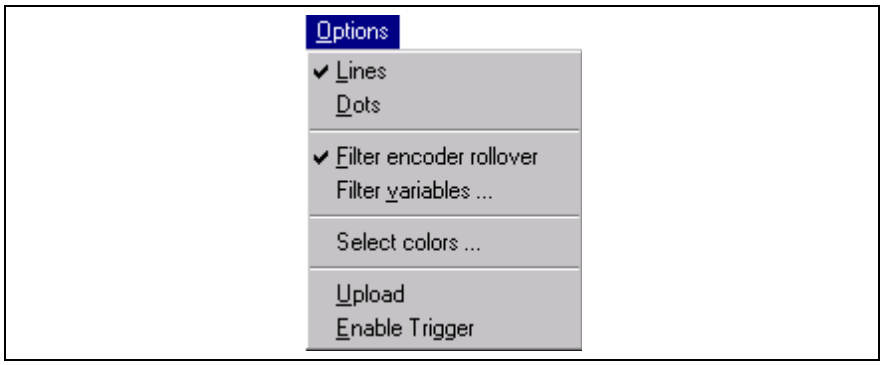


Fig. 4-19: Options Menu

Filter encoder rollover

The position control loop in servo systems is continuously correcting the position of an encoder when at standstill. This continuous correction in position can cause dithering that will be captured by the oscilloscope. When selected, the **Filter encoder rollover** will eliminate any dithering based on the settings of the **Filter variables …** window in Fig. 4-20.

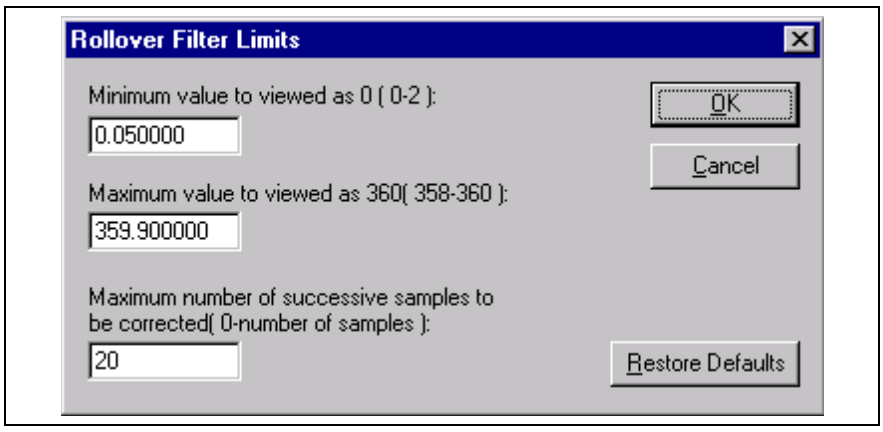


Fig. 4-20: Filter variables

Minimum value to be viewed as 0

A value between 0 and 2 degrees will be used as the filter window for ignoring position dithering. While holding position at 360°, any value between 0 and the minimum value entered will be interpreted as a dither and seen as 360 degrees up to a maximum number of successive samples.

Maximum value to be viewed as 360

A value between 358 and 360 degrees will be used as the filter window for ignoring position dithering. While holding position at 0°, any value between 360 and the maximum value entered will be interpreted as a dither and seen as 0 degrees up to a maximum number of successive samples.

The oscilloscope will immediately capture any position value outside the minimum or maximum filter windows. An example of the filter encoder rollover is shown in Fig. 4-21.

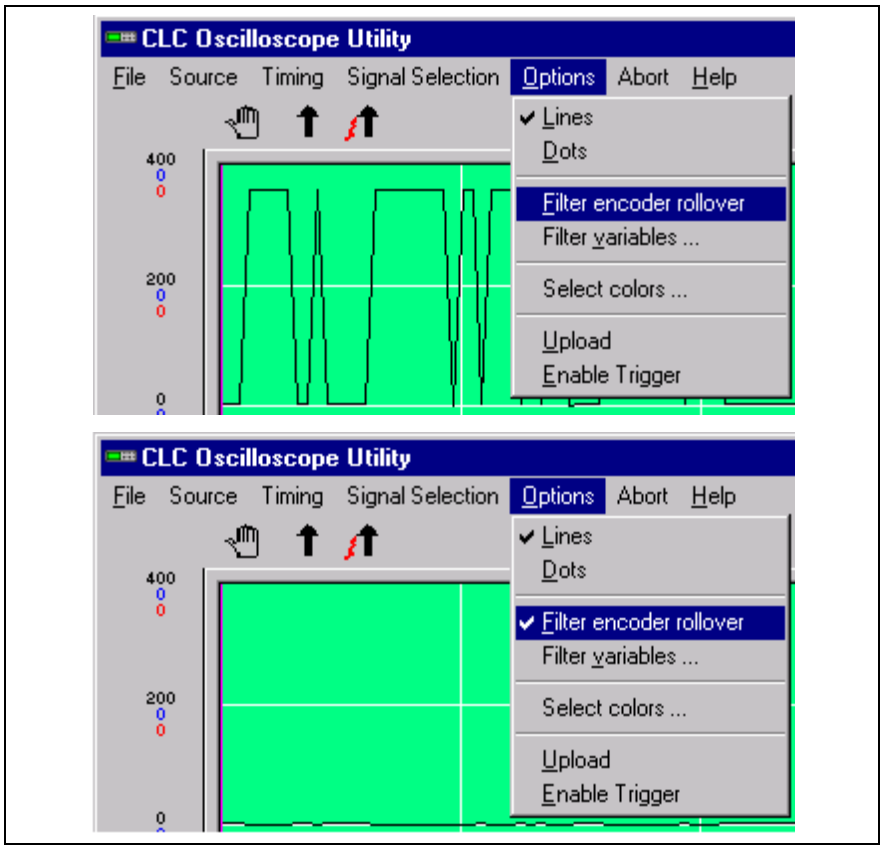


Fig. 4-21: Filter encoder rollover

Select colors…

The three available signal traces are color coded and can be changed for both runtime and memory by selecting a color for each trace in the **Signal color selection** window in Fig. 4-22.

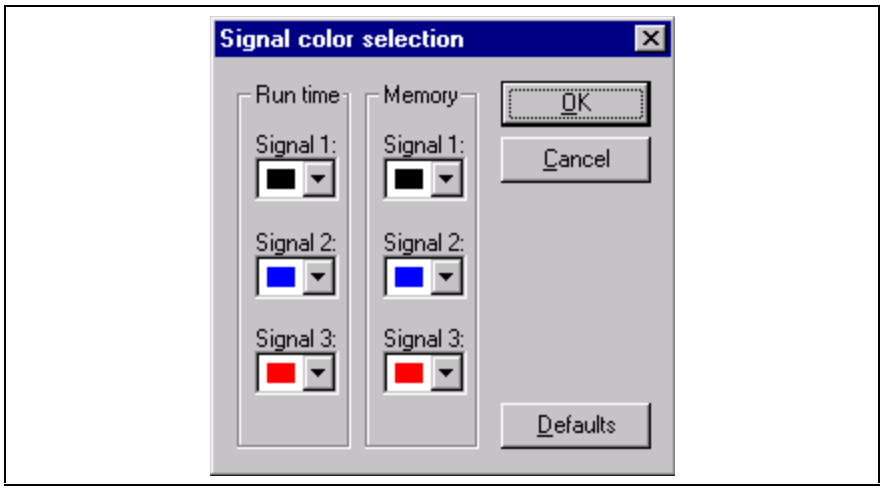


Fig. 4-22: Signal color selection

Abort, Upload and Enable Trigger

The *Abort*, *Upload* and *Enable Trigger* buttons are used to trigger trace captures of configured signals. The **Upload** and **Enable Trigger** functions are also available under the **Options** menu. Furthermore, the **Abort** function

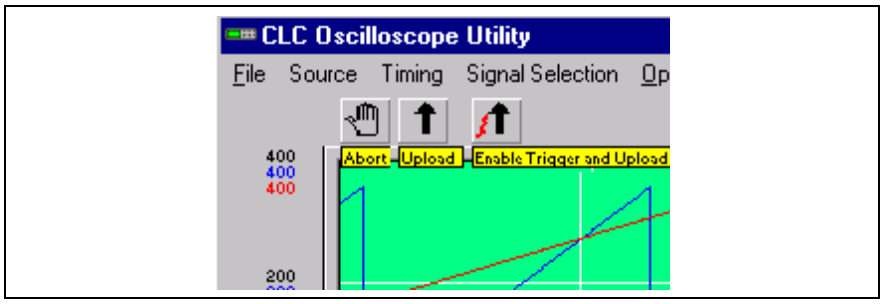


Fig. 4-23: Signal triggering buttons

Oscilloscope memory buttons

Using memory buttons, traces can be stored into memory for viewing and comparing.

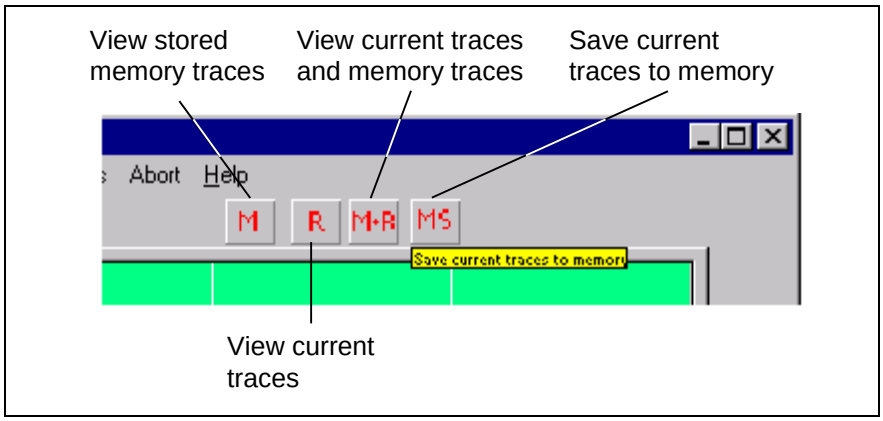


Fig. 4-24: Oscilloscope memory buttons

Manipulating trace signals

When multiple traces are captured at one time, they can be positioned and scaled independently of each other by using the up and down arrows for each trace as shown in Fig. 4-25. The traces can also be turn on or off by clicking on the check boxes to the left of the signal description.

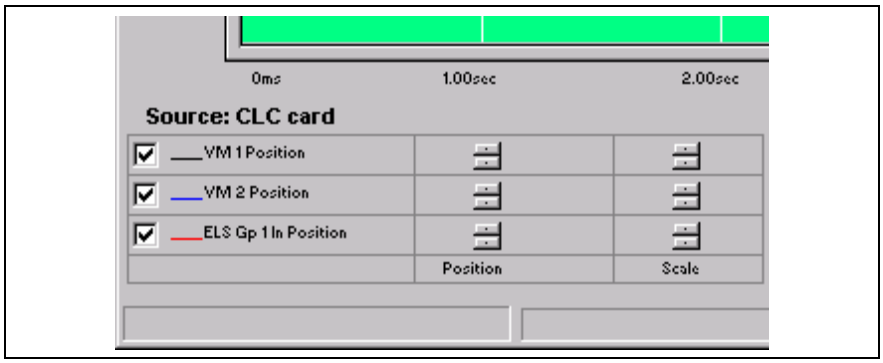


Fig. 4-25: Manipulating trace signal

By positioning the traces above and below each other, the user can more easily distinguish between the signal. Fig. 4-26 shows an example of three traces repositioned for clarity.

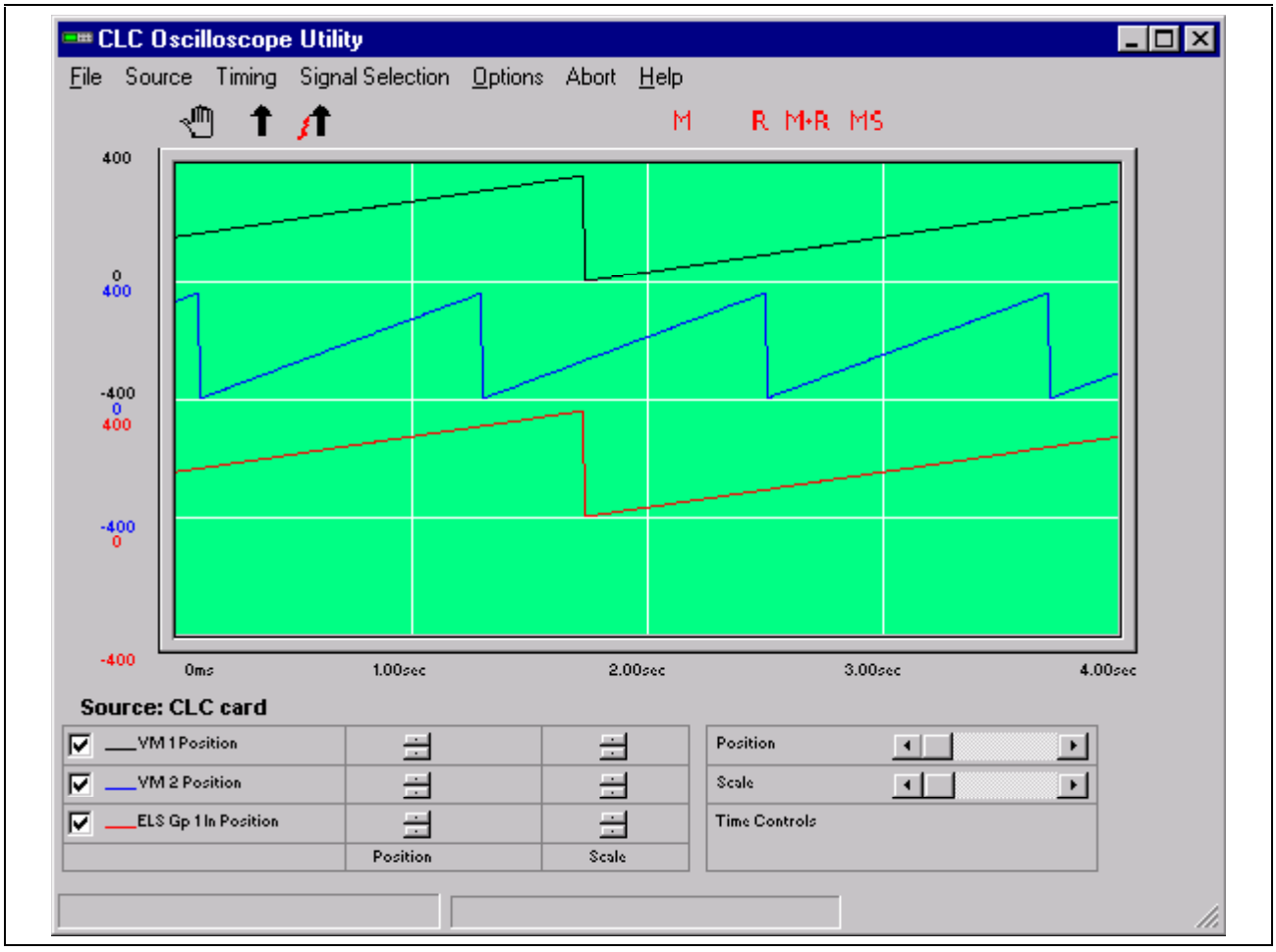


Fig. 4-26: Separated oscilloscope traces

Time Controls

The *Position* and *Scale* functions in Fig. 4-27 are used to more closely analyze a specific area of a captured trace. The *Scale* function acts as a

zoom, allowing the user to view smaller sections of a trace, while the *Position* function controls the horizontal scrolling.

Note: The *Position* function only works after the *Scale* function is used.

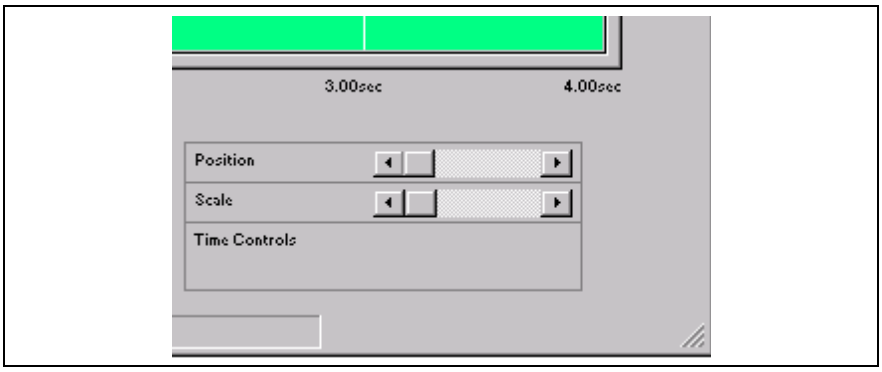


Fig. 4-27: Time Controls

5 CLC TRANSFER UTILITY

5.1 Contents for CLC_XFER Server Help

CLC_XFER is a Microsoft® Windows™ based Dynamic Data Exchange Server application used to transfer data to/from Indramat's CLC / TRANS 01-D motion control cards. Communications is via the CLC_DDE server.

Allowable Data transfer operations

- All(full backup or restore)
- All Card Parameters(system, task, and axis)
- A Drive's parameter set(a drive on the ring)
- An NC Program
- Tool Correction Values

5.2 The Communication Servers Main Window

CLC_XFER window provides access to the help system, transaction log, and about window. During a data transfer, the caption will display the name of the files used in the transfer.

When CLC_XFER is in an icon state, its color will indicate its status.

Red indicates inactive and the last transaction terminated in an error, the message "Failed" is written across the top of the icon.

Yellow indicates the transfer server is being used. While a transfer is in progress, the percent complete is displayed on the top of the icon and a corresponding bar graph across the center. The name of the file involved in the transfer is displayed below the icon.

Green indicates inactive and the last transaction was successful or it was just spawned. The message "Ready" is written across the top of the icon.

5.3 Dynamic Data Exchange Interface

A windows application, known as a *client,* can pass information between other applications known as *servers* using **Dynamic Data Exchange** (**DDE)**. A client establishes a conversation with a server specifying a *Service* and a *Topic*. Once a conversation has been started, a client may request or send information by specifying an *item*.

The service name for the transfer utility is "CLC_XFER".

To transfer data to or from a file

To transfer data use the DDE **EXECUTE** command. The *topic* is the communication type like used for the DDE server(i.e. SERIAL_0). The *item* is an ASCII string containing the direction(R/W), complete path, data descriptor (see above), and program handle (1-10) or drive ID (1-4).

Data transfer operation codes:

a All(full archive or restore)

c Card Parameters(system, task, and axis)

d Drive parameters(a drive on the ring)

To get status or the last error message

To inquire about the status, use the DDE **REQUEST** command. The *topic* is "SERVER" and the *item* is one of the following:

TRANSFER|Status ; returns -2 for error, -1 for ready, 0-100 for percent complete

TRANSFER|ErrorText ; returns last error message

To change a parameter in the server

To change parameters of the transfer utility, use the DDE **POKE** command. The *topic* is “SERVER” and the *item* is one of the following:

TRANSFER|Response_Timeout ; 1-900, set message timeout in seconds

TRANSFER|Display_CLC_Errors ; 0-1, clears/sets if errors are displayed from server

5.4 Activity Log

A record of data transfer transactions is saved to a read only file “XFERLOG.TXT” in the working directory. The file size is limited to 256 lines; the oldest record is deleted to make room for a new entry. Each entry consists of date, time, dde topic, dde item, and status. For failed entries, an indented message describing the failure will be displayed on the following line.

6 Index

4

425 Task B
425 Depth
Probe reading> w3 not zeroed (30) 3-18

5

511 Adaptive Depth Pre-limit Error 3-18
512 Adaptive Depth Part Not Found 3-18

7

779 Adaptive Depth not configured for this axis 3-18
780 Maximum Adaptive Depth feedrate exceeded....3-18
781 Maximum Adaptive Depth deflection exceeded....3-18

A

A02 3-20
A05 3-20
A10 3-20
A11 3-20
Aa11 3-11
Aa12 3-11
Aa14 3-11
Aa20 3-11
Aa21 3-11
Aa22 3-11
About Visual TRANS… 1-43
absolute positioning....3-13
Activity Log 5-2
adaptive depth programming 3-15
Adaptive Depth Control (G08) ..3-15
Adaptive Depth Hardware....3-16
Analog Outputs....4-1
Application Programming Requirements 3-1
Archive… 1-26, 1-43
Automatic Mode....3-2
auxiliary functions 3-32
Auxiliary Functions (NC Code M)3-32
Auxiliary Functions [M] Tab 1-13
Axis....1-10
axis clamping....3-23
Axis Enable and Disable (G20, G21)....3-8

B

Backup....1-26
Backup All Cards....1-30
Backup Card ##....1-28
Basic Homing Program....3-8
Baud Rate....2-5
Block Comment 1-15
Block Jump [J] Tab 1-14
Block… 1-18

C

Card Selection... .... 1-18, 1-37
Check Program Syntax1-15, 1-18, 1-25, 1-43
clamping, axis 3-23
CLC
Back Plane Relay Time-out ... 2-3
Response Time-out 2-3
Server Configuration 2-3
Status Display 2-3
CLC DDE Server 2-2
CLC Serial Ports
Port 1 1-20
Port 2 1-20
CLC Serial Ports.. 1-19
CLC_XFER 5-1
Clear All Forcing 1-35
Clear NC Program… 1-25, 1-43
Comment Tab 1-15
Block Comment 1-15
conditional jump 3-33
Conditional Jump (NC Code JC)3-33
Configuration 1-20
Context Help 1-43
Creating a Program Block.... 1-7

D

Data 1-34
Registers 1-34
DDC 1.1 spindle positioning 3-31
DDE 2-1, 2-4
Conversations 2-5
Dde EXECUTE 5-1
Dde POKE 5-2
Dde REQUEST 5-1
Dde Server 2-1
DDS 2.1 spindle positioning.... 3-31
DDS 2.2 spindle positioning.... 3-31
DDS 3.1 spindle positioning.... 3-31
DDS 3.2 spindle positioning.... 3-31
DeviceNet… 1-21
Diagnostic Log 1-37
DIAX01 spindle positioning.... 3-30
DIAX02 spindle positioning.... 3-31
DIAX03 spindle positioning.... 3-31
DIAX04 spindle positioning.... 3-31
Discrete DEA 28 1-23
Discrete DEA 4 1-23
Discrete DEA 4 and 5 1-23
distance coded linear scale 3-19
DKR spindle positioning.... 3-31
DKS 1.1 spindle positioning.... 3-31
Drive Direction 4-2
Drive Direction Editor 4-2
Drive Homing 4-6
Drive I/O 1-22
Drive I/O…
Discrete DEA 28 1-23
Discrete DEA 4 1-23
Discrete DEA 4 and 5 1-23

Fieldbus....1-23
Interbus-S Configuration 1....1-23
Drive Limits....4-5
Drive Monitoring 4-3, 4-4
Drive Name....4-3
Drive Parameter Editor....4-1
Drive Reference....4-6
Drive Tuning 4-4
Drives Help Directories... 1-23
Drives... 1-23
dwell 3-26
Dwell (NC Code G04 3-26
Dynamic Data Exchange 2-1

E

Edit Bits 1-35
Edit Register....1-35
Enable/Disable Feed To A Positive Stop (G75 & G76)....3-14
end of program 3-1
Error Handling 2-4

F

feed to a positive stop....3-14
feedback devices, external
distance coded linear scale ..3-19
feedrate 3-26
Feedrate (NC Code F)....3-26
Fieldbus....1-23
File
New 1-17
File Menu....1-16
Finishing 1-9
first positioning 3-1
Force a Register....1-35
Format....1-35
Function Tab....1-8
Axis....1-10
Finishing 1-9
Positioning....1-9
Rotary Direction....1-9

G

G00.... 1-10, 3-12
G01.... 1-10, 3-12
G04....3-26
G08....3-15
G20
axis clamping....3-24
G21
axis clamping....3-24
G36....3-21
G37....3-21
G38....3-22
G61....3-13
G62....3-13
adaptive depth control 3-17
jumps....3-33
programming 3-15
G69....3-10
G74.... 1-10, 3-9
distance coded linear scale ..3-19
rotary operation 3-20
G75
programming 3-14
G76
programming.... 3-14
G90.... 3-13
rotary operation.... 3-21
G91.... 3-13
rotary operation.... 3-21

H

HDD spindle positioning 3-31
HDS spindle positioning.... 3-31
Help 1-43
About Visual TRANS.... 1-43
homing.... 3-9
positive stop.... 3-10
positive stop parameters.... 3-11
zero offset.... 3-9

I

incremental positioning.... 3-13
Installing Visual TRANS.... 1-1
Interbus-S Configuration 1.... 1-23
Item Name 2-2

J

JC 3-33
JN 3-33
JR 3-34
JReturn 3-36
JS.... 3-34
JU 3-35
jump
conditional.... 3-33
reverse vector 3-34
unconditional.... 3-33
Jump And Stop 3-34
Jump And Stop (NC Code JS). 3-34
Jump on Event.... 3-35
Jump to Block 000 and Stop program termination.... 3-1
Jump To Subroutine 3-36
Jump to Subroutine JU 3-35

K

KDA spindle positioning.... 3-30

L

Lag Finishing 3-13

M

Manual Mode 3-4
reverse vector 3-34
Maximum Model Deviation.... 4-4
Mechanical.... 4-9
Menu Bar 1-16
Data 1-34
File 1-16
Setup 1-18, 1-37
Status.... 1-37
Tools 1-23
View 1-17
Modes 1-38
Monitoring 4-4
Multi-turn Encoder 4-6

N

NC Code Descriptions 3-7
NC Program Block Number1-8, 1-11
New 1-17
Next Error 1-18

O

Options 1-35
  Clear All Forcing 1-35
  Edit Bits 1-35
  Edit Register 1-35
  Force a Register 1-35
Oscilloscope 4-12
Overview.....4-10

P

Port 1 1-20
Port 2 1-20
Position.....1-12
positional move.....3-12
positioning
  absolute 3-13
  incremental 3-13
  spindle 3-30
Positioning.....1-9
Positioning (NC Code G00, G01, G90 & G91) 3-12
Positive Stop
  homing.....3-10
power loss
  reverse vector.....3-35
Previous Error.....1-18
Program
  Save 1-16
Program Assistant ... 1-7, 1-24, 1-42
  Auxiliary Functions [M] Tab ..1-13
  Block Jump [J] Tab 1-14
  Comment Tab 1-15
  Function Tab.....1-8
  Program Block Tab.....1-8
  Spindle Speed, or Position [S, P] Tab 1-12
  Tool Correction Tab 1-11
Program Block..... 1-15, 1-18
  Create.....1-7
Program Block Number 1-8, 1-11
Program Block Tab.....1-8
Program Download… 1-24, 1-43
program start 3-1
program termination 3-1
Program Upload… 1-24, 1-43
Programming.....3-1
Programming Styles with Example 3-2

R

RAC spindle positioning 3-30
Reference Position
  homing.....3-11
registers
  tool correction..... 3-26, 3-27
Registers 1-34
  Format 1-35
  Options 1-35
  Settings..... 1-37
Restore 1-26
Restore Card ## 1-32
reverse vector 3-34
  auxiliary functions 3-32
  homing 3-9
Reverse Vector (NC Code JR). 3-34
Rotary Direction 1-9
RS485 Converter 2-5
Running Visual TRANS 1-5

S

S-0-0115 Position Feedback Type
  distance coded linear scale . 3-19
S-0-0118 Resolution of Linear Feedback
  distance coded linear scale . 3-19
S-0-0165 Distance-coded Reference Dimension 1
  distance coded linear scale . 3-19
S-0-0166 Distance-coded Reference Dimension 2
  distance coded linear scale . 3-19
S-0-0178 Absolute Offset 2
  distance coded linear scale . 3-19
Saving an NC Program..... 1-16
Serial Communications..... 2-4
Serial Event 2-5
Serial Port..... 2-5
SERVER
  Topic Name 2-7
Service Name 2-1
Settings..... 1-37
Setup 1-18, 1-37
  Card Selection... ..... 1-18, 1-37
  CLC Serial Ports... 1-19
  Configuration 1-20
  DeviceNet... 1-21
  Drive I/O… 1-22
  Drives Help Directories 1-23
  Drives..... 1-23
Smoothing Time Constant 4-4
speed..... 3-12
  spindle 3-30
Speed 1-12
spindle positioning 3-30
  DIAX01 3-30
Spindle Positioning Control (NC Code P)..... 3-30
Spindle speed..... 3-30
Spindle Speed Control (NC Code S) 3-30
Spindle Speed, or Position [S, P] Tab..... 1-12
  Position..... 1-12
  Speed 1-12
start of program 3-1
Status..... 1-37
  Diagnostic Log..... 1-37
  Modes..... 1-38
  System..... 1-41
System..... 1-41

T

Target Block..... 1-14
TDA spindle positioning..... 3-30
Tool Bar 1-42
tool correction 3-26

external transfer.....3-27
program block.....3-27
Tool Correction [T] Tab.....1-11
Tool Corrections (NC Code T)..3-26
Tools.....1-23
Archive..... 1-26, 1-43
Check Program Syntax1-15, 1-18, 1-25, 1-43
Clear NC Program... 1-43
Clear NC Program….. 1-25, 1-43
Program Assistant1-7, 1-24, 1-42
Program Download... 1-43
Program Download… . 1-24, 1-43
Program Upload... 1-43
Program Upload… 1-24, 1-43
Topic Name 2-1

U

unconditional jump 3-33
Unconditional Jump (NC Code JN) 3-33

V

Velocity limit.....4-5
Velocity Loop Integral Action Time 4-5
Velocity Loop Proportional Gain .4-4
View.....1-17
Block… 1-18
Next Error..... 1-18
Previous Error..... 1-18
Visual Trans
Parameters Menu
Drive Monitoring..... 4-3
Visual TRANS
Parameters Menu
Analog Outputs..... 4-1
Drive Homing 4-6
Drive Reference..... 4-6
Drive Tuning 4-4
Mechanical..... 4-9
Visual TRANS
Parameters Menu
Drive Direction 4-2
Drive Limits..... 4-5
Drive Name..... 4-3
Overview..... 4-10

W

With / Without Lag During Positioning (G61 & G62)..... 3-13
Without Lag Finishing..... 3-13
jumps 3-33

Z

zero offset
homing..... 3-10

7 Customer Service Locations

USA

USA ☒ SALES ☒ Service Mannesmann Rexroth Corporation Rexroth Indramat Division 5150 Prairie Stone Parkway USA – Hoffman Estates, IL 60192-3707 Tel.: +1-847-645-3600 Fax: +1-847-645-6201 e-mail: service@indramat.com	USA ☒ SALES ☒ Service Mannesmann Rexroth Corporation Rexroth Indramat Division Central Region Technical Center USA - Auburn Hills, MI 48326 Tel.: +1-248-393-3330 Fax: +1-248-393-2906	USA ☒ SALES ☒ Service Mannesmann Rexroth Corporation Rexroth Indramat Division Southeastern Technical Center 3625 Swiftwater Park Drive USA – Suwanee Georgia 30174 Tel.: +1-770-932-3200 +1-770-932-1903	USA ☒ SALES ☒ Service Mannesmann Rexroth Corporation Rexroth Indramat Division Northeastern Technical Center 99 Rainbow Road USA - East Granby, Connecticut 06026 Tel.: +1-860-844-8377 +1-860-844-8595
USA ☒ SALES ☒ Service Mannesmann Rexroth Corporation Rexroth Indramat Division Charlotte Regional Sales Office 14001 South Lakes Drive USA - Charlotte, North Carolina 28273 Tel.: +1-704-583-9762 +1-704-583-1486			USA Service HOTLINE **+1-800-860-1055** - 7 days / 24hrs -

Service agencies in the USA -
Kundenbetreuungsstellen in den USA

Germany - Deutschland

vom Ausland: (0) nach Landeskennziffer weglassen!!
from abroad: don't dial (0) after country code!

Vertriebsgebiet Mitte ☒ SALES Germany Central ☒ Service Rexroth Indramat GmbH Bgm.-Dr.-Nebel-Str. 2 D - 97816 Lohr am Main Tel.: +49 (0)9352/40-0 Fax: +49 (0)9352/40-4885	Vertriebsgebiet Mitte ☒ SALES Germany Central ☐ Service Mannesmann Rexroth AG Gesch.ber. Rexroth Indramat Lilistraße 14-18 D - 63067 Offenbach Tel.: +49 (0) 69/82 00 90-0 Fax: +49 (0) 69/82 00 90-80	Vertriebsgebiet Ost ☒ SALES Germany East ☒ Service Rexroth Indramat GmbH Beckerstraße 31 D - 09120 Chemnitz Tel.: +49 (0)371/35 55-0 Fax: +49 (0)371/35 55-333	Vertriebsgebiet Ost ☒ SALES Germany East ☐ Service Mannesmann Rexroth AG GB Rexroth Indramat GmbH Holzhäuser Str. 122 D - 04299 Leipzig Tel.: +49 (0)341/86 77-0 Fax: +49 (0)341/86 77-219
Vertriebsgebiet Süd ☒ SALES Germany South ☐ Service Rexroth Indramat GmbH Ridlerstraße 75 D - 80339 München Tel.: +49 (0)89/540138-30 Fax: +49 (0)89/540138-10 e-mail: indramat.mue@t-online.de	Gebiet Südwest ☒ SALES Germany South-West ☒ Service Mannesmann Rexroth AG Vertrieb Deutschland – VD-BI Geschäftsbereich Rexroth Indramat Regionalzentrum Südwest Ringstrasse 70 / Postfach 1144 D - 70736 Fellbach / D - 70701 Fellbach Tel.: +49 (0)711/57 61–100 Fax: +49 (0)711/57 61–125	Vertriebsgebiet Nord ☒ SALES Germany North ☐ Service Rexroth Indramat GmbH Kieler Straße 212 D - 22525 Hamburg Tel.: +49 (0)40/85 31 57-0 Fax: +49 (0)40/85 31 57-15	Vertriebsgebiet Nord ☒ SALES Germany North ☐ Service Mannesmann Rexroth AG Vertriebsniederlassung Region Nord Gesch.ber. Rexroth Indramat Walsroder Str. 93 D - 30853 Langenhagen Tel.: +49 (0) 511/72 66 57-0 Fax: +49 (0) 511/72 66 57-93
Vertriebsgebiet West ☒ SALES Germany West ☒ Service Mannesmann Rexroth AG Vertrieb Deutschland Regionalzentrum West Borsigstrasse 15 D - 40880 Ratingen Tel.: +49 (0)2102/409-0 Fax: +49 (0)2102/409-406		SERVICE - Hotline - 7 Tage / 24h - **HELPDESK** Mon – Fri von 7 – 17 Uhr from 7 a.m. – 5 p.m. Fax: +49 (0)9352/40-4941 Tel.: +49 (0)9352/40- Bernard A. -4894 Kolb R. -4922 Roeper P. -4359 Scheiner W. -4921 **HOTLINE** von 17 - 07 und Sa/So from 5 p.m. – 7 a.m. and Sat/Sun Tel.: +49 (0)172/660 04 06 oder/or Tel.: +49 (0)171/333 88 26	ERSATZTEIL / PARTS Hotline ♦ nur an Werktagen - only on working days - ♦ von 15 -18 Uhr - from 3 – 6 p.m. **Tel. +49 (0) 93 52/40 42 22**

Service agencies in Germany - Kundenbetreuungsstellen in Deutschland

Europe - Europa

vom Ausland: (0) nach Landeskennziffer weglassen, 0 nach Landeskennziffer mitwählen!
from abroad: don't dial (0) after country code, dial 0 after country code!

Austria ☒ SALES ☐ Service Mannesmann Rexroth Ges.m.b.H. Gesch.ber. Rexroth Indramat Hägelingasse 3 A - 1140 Wien Tel.: +43 (0)1/9852540-400 Fax: +43 (0)1/9852540-93	Austria ☒ SALES ☐ Service Mannesmann Rexroth G.m.b.H. Gesch.ber. Rexroth Indramat Industriepark 18 A - 4061 Pasching Tel.: +43 (0)7221/605-0 Fax: +43 (0)7221/605-21	Belgium ☒ SALES ☒ Service Mannesmann Rexroth N.V.-S.A. Gesch.ber. Rexroth Indramat Industrielaan 8 B-1740 Ternat Tel.: +32 (0)2/5830719 Fax: +32 (0)2/5830731 e-mail: indramat@rexroth.be	Denmark ☒ SALES ☒ Service BEC AS Zinkvej 6 DK-8900 Randers Tel.: +45 (0)87/11 90 60 Fax: +45 (0)87/11 90 61
Chechia ☒ SALES ☐ Service Mannesmann-Rexroth, spol.s.r.o. Hviezdoslavova 5 CS - 627 00 Brno Tel.: +420 (0)5/48 126 358 Fax: +420 (0)5/48 126 112	England ☒ SALES ☒ Service Mannesmann Rexroth Ltd. Rexroth Indramat Division Broadway Lane, South Cerney GB - Cirencester, Glos GL7 5UH Tel.: +44 (0)1285/863000 Fax: +44 (0)1285/863030	Finland ☒ SALES ☐ Service Rexroth Mecman Oy Rexroth Indramat division Ansatie 6 SF-017 40 Vantaa Tel.: +358 (0)9/84 91-11 Fax: +358 (0)9/84 91-13 60	France ☒ SALES ☒ Service Mannesmann Rexroth S.A. Division Rexroth Indramat Parc des Barbanniers 4, Place du Village F-92632 Gennevilliers Cedex Tel.: +33 (0)141 47 54 30 Fax: +33 (0)147 94 69 41 Hotline: +33 (0)6 08 33 43 28
France ☒ SALES ☐ Service Mannesmann Rexroth S.A. Division Rexroth Indramat 270, Avenue de Lardenne F - 31100 Toulouse Tel.: +33 (0)5 61 49 95 19 Fax: +33 (0)5 61 31 00 41	France ☒ SALES ☐ Service Mannesmann Rexroth S.A. Division Rexroth Indramat 91, Bd. Irène Joliot-Curie F - 69634 Vénissieux – Cedex Tel.: +33 (0)4 78 78 53 65 Fax: +33 (0)4 78 78 53 62	Italy ☒ SALES ☒ Service Mannesmann Rexroth S.p.A. Divisione Rexroth Indramat Via G. Di Vittoria, 1 I - 20063 Cernusco S/N.MI Tel.: +39 02/92 36 52 70 Fax: +39 02/92 36 55 12	Italy ☒ SALES ☒ Service Mannesmann Rexroth S.p.A. Divisione Rexroth Indramat Via Borgomanero, 11 I - 10145 Torino Tel.: +39 011/7 50 38 11 Fax: +39 011/7 71 01 90
Italy ☒ SALES ☐ Service Mannesmann Rexroth S.p.A. Divisione Rexroth Indramat Via del Progresso, 16 (Zona Ind.) I - 35020 Padova Tel.: +39 049/8 70 13 70 Fax: +39 049/8 70 13 77	Italy ☐ SALES ☒ Service Mannesmann Rexroth S.p.A. Divisione Rexroth Indramat Via Mascia, 1 I - 80053 Castellamare di Stabia NA Tel.: +39 081/8 71 57 00 Fax: +39 081/8 71 68 86	Italy ☒ SALES ☐ Service Mannesmann Rexroth S.p.A. Divisione Rexroth Indramat Viale Oriani, 38/A I - 40137 Bologna Tel.: +39 051/34 14 14 Fax: +39 051/34 14 22	Netherlands ☒ SALES ☐ Service Hydraudyne Hydrauliek B.V. Kruisbroeksestraat 1 (P.O. Box 32) NL - 5281 RV Boxtel Tel.: +31 (0)411/65 19 51 Fax: +31 (0)411/65 14 83 e-mail: indramat@hydraudyne.nl
Netherlands ☐ SALES ☒ Service Hydrocare B.V. Kruisbroeksestraat 1 (P.O. Box 32) NL - 5281 RV Boxtel Tel.: +31 (0)411/65 19 51 Fax: +31 (0)411/67 78 14	Norway ☒ SALES ☐ Service Rexroth Mecman AS Rexroth Indramat Division Berghagan 1 or: Box 3007 N -1405 Ski-Langhus N -1402 Ski Tel.: +47 (0)64 86 41 00 Fax: +47 (0)64 86 90 62	Poland ☒ SALES ☐ Service Mannesmann Rexroth Sp.zo.o. Biuro Poznan ul. Dabrowskiego 81/85 PL - 60-529 Poznan Tel.: +48 061/847 67 99 Fax: +48 061/847 64 02	Russia ☐ SALES ☒ Service Tschudnenko E.B. Arsenia 22 RUS - 153000 Ivanovo Rußland Tel.: +7 093/223 96 33 oder/or +7 093/223 95 48 Fax: +7 093/223 46 01
Spain ☒ SALES ☒ Service Mannesmann Rexroth S.A. Divisiòn Rexroth Indramat Centro Industrial Santiga Obradors s/n E-08130 Santa Perpetua de Mogoda Barcelona Tel.: +34 937 47 94 00 Fax: +34 937 47 94 01	Spain ☒ SALES ☒ Service Goimendi S.A. División Rexroth Indramat Jolastokieta (Herrera) Apartado 11 37 E - 20017 San Sebastian Tel.: +34 9 43/40 01 63 Fax: +34 9 43/39 17 99	Sweden ☒ SALES ☒ Service Rexroth Mecman Svenska AB Rexroth Indramat Division Varuvägen 7 S - 125 81 Stockholm Tel.: +46 (0)8/727 92 00 Fax: +46 (0)8/647 32 77	Slowenia ☒ SALES ☒ Service Rexroth Indramat elektromotorji d.o.o. Otoki 21 SLO - 64 228 Zelezniki Tel.: +386 64/61 73 32 Fax: +386 64/64 71 50
Turkey ☒ SALES ☒ Service Mannesmann Rexroth Hidropar A..S. Fevzi Cakmak Cad No. 3 TR - 34630 Sefaköy Istanbul Tel.: +90 212/541 60 70 Fax: +90 212/599 34 07	Switzerland ☒ SALES -East- ☒ Service Mannesmann Rexroth Schweiz AG Gesch.ber. Rexroth Indramat Gewerbestraße 3 CH - 8500 Frauenfeld Tel.: +41 (0)52/720 21 00 Fax: +41 (0)52/720 21 11	Switzerland ☒ SALES -West- ☐ Service Mannesmann Rexroth Suisse SA Département Rexroth Indramat Rue du village 1 CH - 1020 Renens Tel.: +41 (0)21/632 84 20 Fax: +41 (0)21/632 84 21	

European Service agencies outside of Germany -
Europäische Kundenbetreuungsstellen außerhalb Deutschland

Outside of Europe - Außerhalb Europa

vom Ausland: (0) nach Landeskennziffer weglassen!
from abroad: don't dial (0) after country code!

Argentina ☒ SALES ☐ Service Mannesmann Rexroth S.A.I.C. Division Rexroth Indramat Acassusso 48 41/7 RA - 1605 Munro (Buenos Aires) Tel.: +54 (0)11/4756 01 40 Fax: +54 (0)11/4762 6862 e-mail:mannesmann@impsat1.com.ar	Argentina ☒ SALES ☒ Service NAKASE Servicio Tecnico CNC Calle 49, No. 5764/66 RA - 1653 Villa Balester Prov. - Buenos Aires Tel.: +54 (0) 11/4768 36 43 Fax: +54 (0) 11/4768 24 13 e-mail: nakase@usa.net nakase@infovia.com.ar	Australia ☒ SALES ☒ Service AIMS - Australian Industrial Machinery Services Pty. Ltd. Unit 3/45 Horne ST Campbellfield , VIC 3061 AUS - Melbourne Tel.: +61 (0)3/93 59 02 28 Fax: +61 (0)3/93 59 02 86	Australia ☒ SALES ☐ Service Mannesmann Rexroth Pty. Ltd. No. 7, Endeavour Way Braeside Victoria, 31 95 AUS – Melbourne Tel.: +61 (0)3/95 80 39 33 Fax: +61 (0)3/95 80 17 33 e-mail: mel@rexroth.com.au
Brazil ☒ SALES ☒ Service Mannesmann Rexroth Automação Ltda. Divisão Rexroth Indramat Rua Georg Rexroth, 609 Vila Padre Anchieta BR - 09951-270 Diadema-SP [Caixa Postal 377] [BR-09901-970 Diadema-SP] Tel.: +55 (0)11/745 90 60 +55 (0)11/745 90 70 Fax: +55 (0)11/745 90 50 e-mail: awittwer@rexroth.com.br	Brazil ☐ SALES ☒ Service Mannesmann Rexroth Automação Ltda. Divisão Rexroth Indramat R. Dr.Humberto Pinheiro Vieira, 100 Distrito Industrial BR - 89220-390 Joinville - SC [Caixa Postal 1273] Tel./Fax: +55 (0)47/473 58 33 Mobil: +55 (0)47 974 66 45 e-mail: prochnow@zaz.com.br	Canada ☒ SALES ☒ Service Basic Technologies Corporation Burlington Division 3426 Mainway Drive Burlington, Ontario Canada L7M 1A8 Tel.: +1 905/335 55 11 Fax: +1 905/335-41 84	China ☐ SALES ☒ Service Mannesmann Rexroth (China) Ldt. Shanghai Parts & Service Center 199 Wu Cao Road, Hua Cao Minhang District PRC - Shanghai 201 103 Tel.: +86 21/62 20 00 58 Fax: +86 21/62 20 00 68
China ☒ SALES ☐ Service Mannesmann Rexroth (China) Ldt. 15/F China World Trade Center 1, Jianguomenwai Avenue PRC - Beijing 100004 Tel.: +86 10/65 05 03 80 Fax: +86 10/65 05 03 79	China ☒ SALES ☐ Service Mannesmann Rexroth (China) Ldt. A-5F., 123 Lian Shan Street Sha He Kou District PRC - Dalian 116 023 Tel.: +86 411/46 78 930 Fax: +86 411/46 78 932	Hongkong ☒ SALES ☒ Service Rexroth (China) Ldt. 1/F., 19 Cheung Shun Street Cheung Sha Wan, Kowloon, Hongkong Tel.: +852 22 62 51 00 Fax: +852 27 44 02 78	India ☒ SALES ☒ Service Mannesmann Rexroth (India) Ltd. Rexroth Indramat Division Plot. 96, Phase III Peenya Industrial Area IND - Bangalore - 560058 Tel.: +91 (0)80/8 39 73 74 Fax: +91 (0)80/8 39 43 45
India ☒ SALES ☒ Service Mannesmann Rexroth (India) Ltd. Rexroth Indramat Division Plot. A-58, TTC Industrial Area Thane Turbhe Midc Road Mahape Village IND - Navi Mumbai - 400 701 Tel.: +91 (0)22/7 61 46 22 Fax: +91 (0)22/7 68 15 31	Indonesia ☒ SALES ☐ Service PT. Rexroth Wijayakusuma Jl. Raya Bekasi Km 21 Pulogadung RI - Jakarta Timur 13920 Tel.: +62 21/4 61 04 87 +62 21/4 61 04 88 Fax: +62 21/4 60 01 52	Japan ☐ SALES ☒ Service Rexroth Automation Co., Ltd. Service Center Japan Yutakagaoka 1810, Meito-ku, NAGOYA 465-0035, Japan Tel.: +81 (0)52/777 88 41 +81 (0)52/777 88 53 +81 (0)52/777 88 79 Fax: +81 (0)52/777 89 01	Japan ☒ SALES ☒ Service Rexroth Automation Co., Ltd. Rexroth Indramat Division 1F, I.R. Building Nakamachidai 4-26-44, Tsuzuki-ku YOKOHAMA 224-0041, Japan Tel.: +81 (0)45/942 72 10 Fax: +81 (0)45/942 03 41
Mexico ☒ SALES ☐ Service Mannesmann Rexroth Mexico S.A. de C.V. Calle Neptuno 72 Unidad Ind. Vallejo MEX - 07700 Mexico, D.F. Tel.: +52 5 754 17 11 +52 5 754 36 84 +52 5 754 12 60 Fax: +52 5 754 50 73 +52 5 752 59 43 e-mail: gsoria@rexroth-mexico.com	Korea ☒ SALES ☒ Service Mannesmann Rexroth-Korea Ltd. Rexroth Indramat Division 1500-12 Dadae-Dong- Saha-Ku Pusan, 604-050 Republic of South Korea Tel.: +82 (0)51/2600 741 Fax: +82 (0)51/2600 747	Korea ☒ SALES ☒ Service Seo Chang Corporation Ltd. Room 903, Jeail Building 44-35 Yeouido-Dong Yeoungdeungpo-Ku C.P.O.Box 97 56 ROK - Seoul Tel.: +82 (0)2/7 80 82 08 +82 (0)2/7 80 82 09 Fax: +82 (0)2/7 84 54 08	South Africa ☒ SALES ☒ Service TECTRA Automation (Pty) Ltd. 28 Banfield Road,Industria North RSA - Maraisburg 1700 Tel.: +27 (0)11/673 20 80 Fax: +27 (0)11/673 72 69
Taiwan ☒ SALES ☐ Service Rexroth Uchida Co., Ltd. No.1, Tsu Chiang Street Tu Cheng Ind. Estate Taipei Hsien, Taiwan, R.O.C. Tel.: +886 2/2 68 13 47 Fax: +886 2/2 68 53 88			

Service agencies outside of Europe -
Kundenbetreuungsstellen außerhalb Europa

287125

Printed in Germany

Rexroth Indramat